

NASA Technical Memorandum 107248
ICOMP-96-01

Institute for Computational Mechanics in Propulsion (ICOMP)

Tenth Annual Report – 1995

June 1996



National Aeronautics and
Space Administration



Institute for Computational Mechanics in Propulsion (ICOMP)

Tenth Annual Report - 1995

Editors:

Theo G. Keith, Jr.
Karen Balog
Louis A. Povinelli

June 1996

CONTENTS

	Page
INTRODUCTION	1
THE ICOMP STAFF OF VISITING RESEARCHERS	2
RESEARCH IN PROGRESS	3
REPORTS AND ABSTRACTS	31
SEMINARS	43
WORKSHOP ON ALLSPD-3D COMBUSTOR CODE	51
WORKSHOP ON AST ENGINE NOISE	55

INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP)

TENTH ANNUAL REPORT

1995

SUMMARY

The Institute for Computational Mechanics in Propulsion (ICOMP) was formed to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. ICOMP is operated by the Ohio Aerospace Institute (OAI) and funded via a cooperative agreement by the NASA Lewis Research Center in Cleveland, Ohio. This report describes the activities at ICOMP during 1995.

INTRODUCTION

The Institute for Computational Mechanics in Propulsion (ICOMP) was established at the NASA Lewis Research Center in September 1985. The overall purpose was to improve problem-solving capabilities in all aspects of computational mechanics relating to propulsion. ICOMP provides a means for researchers with experience and expertise to spend time in residence at Lewis performing research to improve computational capability in the many broad and interacting disciplines of interest in aerospace propulsion.

The scope of the ICOMP program is to advance the understanding of aerospace propulsion physical phenomena and to improve computer simulation of aerospace propulsion systems and components. The specific areas of interest in computational research include: fluid mechanics for internal flows; CFD methods; turbulence modelling; and computational aeroacoustics.

This report summarizes the activities at ICOMP during 1995. Two very significant events occurred over the past year: (1) ICOMP turned 10 years old and (2) ICOMP expanded operations by adding researchers at Wright Laboratory and by assisting industry with a few problems.

The following sections of this report provide lists of the resident and visiting researchers, their affiliations and educational backgrounds. This section is followed by reports of RESEARCH IN PROGRESS, REPORTS AND ABSTRACTS published over the past year and SEMINARS presented throughout the year. The agendas and overviews of two very productive workshops held in 1995 are also given. The first was a two-day event entitled "ALLSPD-3D Combustor Code", held November 14-15, 1995. The second was a three-day event entitled "AST Engine Noise" held December 12-14, 1995.

THE ICOMP STAFF OF VISITING RESEARCHERS

The ICOMP research staff for 1995 is shown in Table I. A total of twenty-nine researchers were in residence at ICOMP for periods varying from a few days to a year. The resident staff numbered twenty-three while the visiting staff, including one graduate student, numbered six.

As usual the resident researchers were very productive Table II provides a numerical summary of ICOMP during its first nine years in terms of research staff size and technical output as measured by the numbers of seminars, reports and workshops. The relatively small number of seminars reflects, as in 1994, to a large extent, 1) the strong emphasis placed by the turbulence modeling group and the aeroacoustics group on applying their existing capabilities for customer use in the industrial and user communities, and 2) the conducting of the two extensive workshops. These objectives were judged to be of higher priority than the seminars.

Table I. - The ICOMP Research Staff-1995

A. Resident Staff.

Kumud Ajmani, Ph.D., Mechanical Engineering, Virginia Polytechnic Institute and State University, 1991. January, 1992--Present.

Hannes Benetschik, Ph.D., Mechanical Engineering, TU Aachen Institute, 1992. November, 1995--Present.

Tawit Chitsomboon, Ph.D., Mechanical Engineering, Old Dominion University, 1986. January, 1990--September, 1995.

Joongkee Chung, Ph.D., Mechanical Engineering, University of California, Berkeley, 1991. May, 1992--Present.

Datta Gaitonde, Ph.D., Mechanical and Aerospace Engineering, Rutgers University, 1989. September, 1995-- Present.

Ehtesham Hayder, Ph.D., Mechanical and Aerospace Engineering, Princeton University, 1988. February, 1991--September, 1995.

Duane R. Hixon, Ph.D., Aerospace Engineering, Georgia Institute of Technology, 1993. October, 1993--September, 1995.

Bo-nan Jiang, Ph.D., Engineering Mechanics, University of Texas, Austin, 1986. October, 1987--Present.

Kai-Hsiung Kao, Ph.D., Aerospace Engineering Sciences, University of Colorado, 1989. November, 1992--Present.

William W. Liou, Ph.D., Aerospace Engineering, Pennsylvania State University, 1990. November, 1990--Present.

James M. Loellbach, Ph.D. expected 1995, Aeronautical and Astronautical Engineering, University of Illinois. May, 1992 -- Present.

David Modiano, Ph.D., Computational Fluid Dynamics, Massachusetts Institute of Technology, 1993. February, 1993--January, 1995.

Andrew Norris, Ph.D., Mechanical and Aerospace Engineering, Cornell University, 1993. June, 1993--Present.

Aamir Shabbir, Ph.D., Mechanical Engineering, State University of New York, Buffalo, 1987. June, 1991--Present.

Shyue-Horng Shih, Ph.D., Aerospace Engineering, University of Cincinnati, 1993. June, 1993--Present.

Tsan-Hsing Shih, Ph.D., Aerospace Engineering, Cornell University, 1984. March, 1990--Present.

Erlendur Steinthorsson, Ph.D., Mechanical Engineering, Carnegie Mellon University, 1992. January, 1992--Present.

Gerald Trummer, B.S., Michigan State University, 1982. September, 1995--Present.

Fu-Lin Tsung, Ph.D. expected 1995, Aerospace Engineering, Georgia Institute of Technology. March, 1993--Present.

Wu, Jie, Ph.D., Civil Engineering, University College of Swansea, 1993. March, 1993--Present.

Zhigang Yang, Ph.D., Mechanical and Aerospace Engineering, Cornell University, 1989. July, 1990--December 1995.

Shaye Yungster, Ph.D., Aeronautics and Astronautics, University of Washington, 1989. November, 1989--Present.

Jiang Zhu, Ph.D., Mechanics, Institut National Polytechnique de Grenoble, 1986. April, 1992--Present.

B. Visiting Staff/Consultants.

Thomas Hagstrom, Ph.D., Applied Mathematics, California Institute of Technology, 1983. Associate Professor, Department of Mathematics and Statistics, University of New Mexico.

S. I. Hariharan, Ph.D., Applied Mathematics, Carnegie Mellon University, 1980. Professor, Department of Mathematical Sciences, University of Akron.

Richard H. Pletcher, Ph.D., Mechanical Engineering, Cornell University, 1966. Professor, Department of Mechanical Engineering, Iowa State University.

Gretar Tryggvason, Ph.D., Engineering, Brown University, 1985. Associate Professor, Department of Mechanical Engineering and Applied Mechanics, University of Michigan.

Eli Turkel, Ph.D., Applied Mathematics, New York University, 1970. Professor, Department of Mathematics, Tel Aviv University, Tel Aviv, Israel.

C. Graduate Students.

Scott A. Dudek, Department of Mechanical Engineering, University of California, Berkeley.

Table II. - ICOMP STATISTICS (1986 TO 1995)

	1986	1987	1988	1989	1990	1991	1992	1993	1994	1995
RESEARCHERS	23	43	50	46	47	49	58	64	50	30
SEMINARS	10	27	39	30	37	26	32	46	3	15
REPORTS	2	9	22	32	25	29	27	51	32	28
WORKSHOP/ LECT. SERIES	1	0	2	1	1	1	1	1	2	2
NO. OF PRESENTATIONS	7	0	21	14	15	21	15	33	40	23

Research in Progress

RESEARCH IN PROGRESS

KUMUD AJMANI

Research Area: Development of Codes for Parallel Processing

Work continued towards the development of an efficient Navier-Stokes code for CFD simulations on Parallel Machines. Most of the research was done in close collaboration with Dr. Meng-Sing Liou of NASA Lewis. The primary goal of this work is to support the HPCC program at NASA Lewis by developing a parallelized Navier-Stokes CFD code in order to perform evaluations of HSCT components like inlets, compressors, combustors, turbines etc. The code aims to be portable across different parallel computing environments, particularly among clusters of workstations, the Cray T3D and the IBM SP2. The code also aims to be generic enough to accommodate newer architectures and include state-of-the-art programming techniques to ensure acceptance by industry and end-users.

A three-dimensional Navier-Stokes code has been ported from a single-processor environment to a distributed-memory multi-processor environment. The code is designed to easily incorporate the finest details of the underlying geometry into the CFD simulation. This is accomplished by generating individual, overlapping (CHIMERA) mesh for each component or sub-component. The strategy to parallelize the CFD code has been performed at two levels. First, a coarse-grained distribution of work among processors has been implemented. This translates to the assignment of one mesh geometry per processor, with each processor functioning independently on its set of data. Secondly, a fine-grained distribution of work is attempted for larger meshes, i.e., the data on each large mesh is divided equally into the number of remaining processors. This second step helps in balancing the workload across processors.

The parallel code has been tested on the 64-CPU Cray T3D and the 32-node LACE cluster at NASA Lewis. The first test case has been the computation of secondary flow through a PW4000 compressor. The flow system is modeled with 15 separate meshes and each mesh is assigned to one processor on the Cray T3D. The processing time on the T3D is found to be 25% faster than the Cray YMP (which is the fastest serial machine at Lewis). The processing time on the LACE cluster is twice as fast as the T3D. It should be noted that the largest mesh is 50 times larger than the smallest mesh for this test case, which leads to highly unbalanced processing loads among the 15 processors.

HANNES BENETSCHIK

Research Area: Turbomachinery and Grid Generation Codes

Since November 1995, when Dr. Benetschik began his affiliation with ICOMP, his research work has involved a simulation of a turbine rotor-stator interaction in three dimensional space. Two different codes are to be employed: the code of Chunill Hah of NASA Lewis and the code Dr. Benetschik developed with his former co-workers at the Technical University at Aachen, Germany.

The first stage of Dr. Benetschik's work at ICOMP was concerned with the application of the grid generating code system of James Loellbach of ICOMP, where an I-topology is employed, which is known to exhibit a geometric discontinuity across a defined mid-blade-passage-line. During a subsequent stage, Dr. Benetschik's work was focussed on an O-type grid generator, which retains the metric

continuity, except for the centerline of the internal O-type block, which is used within the tip clearance gap.

Using both grid generators, grid systems for a 1+1/2-stage turbine have been generated for the two flow solvers to be employed. The present work involves the refinement and an extension of the flow solver initiated at the Aachen.

TAWIT CHITSOMBOON

Research Area: Code Development for Nozzle Flows

This years' efforts involved completing the development of a wall function $K-\varepsilon$ model for the NPARC Navier-Stokes code. This turbulence model was being implemented in NPARC to improve the capability of the code to calculate mixing flow fields, such as those found in mixer-ejector nozzles. A unique part of this work was modifying the artificial dissipation scheme near no-slip boundaries in order to avoid nonphysical damping in such regions. The turbulence model was validated against several cases including a flat plate, a backward facing step, an ejector nozzle, and a nozzle that was tested as part of the High Speed Research (HSR) Program. A report documenting the development and validation of this work has been initiated.

JOONGKEE CHUNG

Research Area: Code Development for Unsteady Inlet Flows Using Parallel Processing

Subroutines developed for time accurate computations involving Controls/CFD interdisciplinary research which included a compressor face boundary condition, upstream perturbations, shock tracking, pressure monitoring, multi-block CFL number formulation and dimensional timing were incorporated into the latest version of the NPARC code.

Time-accurate computations to study inlet transients continued and the Mach number base compressor face boundary condition proved to be nonoscillatory and more representative of the characteristics at the engine face for both inviscid and viscous cases. Viscous validation using the new boundary condition required a large amount of CPU time, which lead to the development of a new algorithm. This modified implicit scheme uses a subiteration technique to produce 2nd-order time accuracy without increasing the CPU requirement for large matrix inversion. The subiteration technique allowed a larger time step with a corresponding reduction of CPU time for many cases.

2-D and 3-D CFD run data using various supersonic inlets were delivered to the controls research team for continued model reduction studies. Development and improvements of 2-D parallel NPARC was performed in cooperation with Indiana Univ. Purdue Univ. Indianapolis (IUPUI). This parallel code proved to be efficient in CPU reduction when the computation was performed on the Lewis LACE cluster.

The following represent the research goals for FY96:

1. Implement unsteady modifications into the official version of NPARC code.
2. Develop a new bleed and bypass boundary condition.
3. Validate and assess the capabilities of the viscous version of improved compressor face boundary condition and implicit subiteration technique.

4. Couple two codes (NPARC and ADPAC) for a high-speed inlet study in collaboration with Dr. Ambady Suresh at NYMA.
5. Perform viscous computations using the parallel NPARC 2-D code.
6. Deliver CFD data for high-speed inlet calculations to the controls team for model reduction studies.

DATTA GAITONDE

Research Area: Shock Boundary Layer Interaction, High Order Schemes for Computational Electromagnetics

This work examines numerically viscous-inviscid phenomena in shock-wave/ turbulent boundary layer interactions. Of particular interest is the relationship between the kinematic structure and features of engineering interest, specifically surface shear stresses and heat transfer rates. The full mean compressible Navier-Stokes equations are solved with turbulence effects being incorporated through several eddy-viscosity based models of varying complexity. Numerical issues addressing simulation integrity are carefully examined. Post-processing techniques employ unique approaches to identification and analysis of three-dimensional coherent structures. For both fin-based and cylinder/offset flare geometries, good agreement is obtained with experiment for surface pressure, shock structure and surface streamlines. Discrepancies in skin friction and heat transfer are however noted, mostly in regions of large cross flow where "extra strain rates" violate the validity of the eddy viscosity assumption. The flow structure in double-fin geometries is described in terms of a separated boundary layer, vortex interaction flow, symmetric centerline vortices and entrainment flow. In the flare based geometry, a horseshoe type of vortical structure is formed near the upper symmetry plane. The legs of this structure wrap around the juncture and turn streamwise near the lower symmetry plane.

Work is also progressing on the development and application of high order schemes for computational electro-magnetics (CEM). The finite volume schemes employ concepts from the essentially non-oscillatory (ENO) formulation with the primitive function approach. The resulting schemes are being compared with the baseline third order upwind-biased method implemented in prior work within the framework of order of accuracy as well as Fourier analysis of dispersion and anisotropy errors. In an effort to characterize practical issues, implementation has focussed thus far on multi-dimensional canonical problems on highly curvilinear mesh systems.

THOMAS HAGSTROM

Research Area: Algorithms for Boundary Layer Value Problems, Domain Decomposition

Research work during FY95 involved three separate efforts:

1. Boundary Conditions for Unsteady Compressible Flow Simulations

Collaboration with J. Goodrich (NASA), S. I. Hariharan (ICOMP), and E. Hayder (ICOMP) on boundary conditions for hyperbolic systems based both on geometrical optics and progressive wave expansions continued. The primary applications are to the linearized compressible Euler equations and to Maxwell's equations.

The sixth order conditions based on geometrical optics have been very successfully used by Goodrich on a subset of the NASA benchmark problems in computational aeroacoustics, where in fact they produced the most accurate solutions to date. Currently plans are being developed for a series of tests for problems in duct acoustics. A barrier to their use for exterior problems is the treatment of corners, an issue which is being addressed. The idea here is to derive compatibility conditions guaranteeing the regularity of the solution there. Derivation of the boundary conditions to parallel flows (to low order) has also been extended, and Hayder has implemented a second order version in his unsteady jet code. Finally, exponential convergence was proven as the order of the conditions is increased for the simple case of the wave equation in a half-space. The numerical experiments will also be used to assess the sharpness of these estimates.

In joint work with Hariharan, second order conditions based on progressive wave (far-field) expansions have been derived and implemented in a 2-4 MacCormack code. They have been applied both to the simulation of sound produced by a quadrupole source, propagated through a vortex dipole, and to the gust problem, the sound produced by a vortex impinging on a flat plate. Wake corrections were also constructed to take account of the presence of vorticity at outflow. The use of second order conditions substantially improves the accuracy in comparison with first order or characteristic conditions. Currently consideration is being given to the development of precise error estimates and their verification by numerical experiment.

2. Simulations of Reacting Flows in the Zero Mach Number Limit

Work continued with Dr. K. Radhakrishnan of NYMA on a high-order numerical algorithm for simulating unsteady and steady reacting flows in the zero Mach number asymptotic limit. Recently, the steady problem has been emphasized. Improvements in the time-stepping scheme and the mesh adaptivity have led to both better efficiency and robustness of the code. Computations of hydrogen-oxygen and methane flames have been carried out for various pressures and temperatures, demonstrating the method's capability to solve the problem using only a generic initial approximation. Convergence acceleration via Newton-like methods, in particular the recursive projection method and Broyden's method were examined. A study of the eigenvalues of the Jacobian has been made in the hopes of gaining better insight into the observed behavior.

Next attention will be directed towards unsteady problems, such as ignition, flame in vessels, and flame instabilities. The properties of the developed time stepping scheme, which is a linearly implicit W-method based on the extrapolation to high order of an Euler-type method will be analyzed. Interesting stability problems require the extension of the code to two space dimensions. The primary barrier to this is the extension of a previously developed adaptive meshing technique.

3. High-Order Schemes for Wave Propagation

Work was also initiated on the analysis and extension of Goodrich's high-order methods for linear wave propagation. For problems with constant (piecewise constant) coefficients, these methods are most likely optimal from the point of view of flop counts and memory requirements. Their variable coefficient extensions require no additional work, but do need extra storage. Questions under consideration include the development of a stability theory (their excellent stability properties are evident from the numerical experiments), extensions to nonrectangular meshes, applications to Maxwell's equations, and experiments with variable coefficient systems, such as the linearized Euler

equations for shear flow profiles. The theoretical issues hinge on the study of multidimensional polynomial interpolation.

S. I. HARIHARAN

Research Area: Applied Mathematics, Numerical Solution of Partial Differential Equations, Acoustics, Electrodynamics

During this period, further progress on "Sound Radiation from thin Airfoils" with J. R. Scott of NASA Lewis was made. The paper in progress on this subject was improved by incorporating new results. Also, work on linearized Euler equations was continued.

M. EHTESHAM HAYDER

Research Area: Computation of Jet Noise in the Source Region Near the Nozzle Exit

During the past year at ICOMP, research was concentrated mainly in two subareas. They are, (a) efficient implementation of the numerical models in parallel computing platforms and (b) development and evaluation of numerical models. In the first subarea, we evaluated various parallel computing platforms [1-4], such as the Cray YMP, the Cray T3D, the IBM- SP2 and a cluster of workstations, etc. to study the scalability and communication issues related to large scale computations. One of the focus of this study was to evaluate various networks like ATM, Ethernet, FDDI, etc. and architectures for distributed computations on network of workstations. Evaluations of available software tools for parallel computations were also made. A jet code [5] for axisymmetric flows for parallel computation studies was used. In addition, a collaborative effort with Erlendur Steinthorsson was initiated to study parallel computations of a multiblock finite volume code (TRAF3D) for turbine cooling.

In the second subarea, several non-reflecting boundary conditions for jet flow calculations [6,7] were evaluated. Formulation of the boundary conditions is an important element in a numerical model. Reflections at boundaries are common in many numerical simulations. These reflections contaminate the numerical solution inside the computational domain. In addition to examining many popular boundary conditions, issues relating to stretching and filtering near the outflow were also addressed. A boundary condition [8] was developed and the effectiveness of various artificial dissipation models [9] was examined for aeroacoustic computations.

References

- [1] Hayder, M. E., Jayasimha, D. N., and Pillay, S. K., (1995), "Parallel Navier-Stokes Solutions on Shared and Distributed Memory Architectures", AIAA paper 95-0577.
- [2] Hayder, M. E., Jayasimha, D. N., and Pillay, S. K., (1995), "A Scalability and Communication Study on Parallel Computers for Jet Noise Computations", Presented in the Computational Aerosciences Workshop, NASA Ames Research Center, Moffett Field, CA, March 1995.
- [3] Jayasimha, D. N., Hayder, M. E., and Pillay, S. K., (1995), "Parallelizing Navier-Stokes Computations on a Variety of Architectural Platforms", Presented in Supercomputing'95, San Diego, December 1995.
- [4] Hayder, M. E. and Jayasimha, D. N., (1995), "Navier Stokes Simulations of Jet Flows on a Network of Workstations", To appear in the AIAA Journal.

- [5] Hayder, M. E., Turkel, E. and Mankbadi, R. R. (1993), "Numerical Simulation of a High Mach Number Jet", AIAA paper 93-0653, NASA TM 105985.
- [6] Hayder, M. E. and Turkel, E., (1995), "Nonreflecting Boundary Conditions for Jet Flow Computations", AIAA Journal, Vol. 33, No. 12, pp 2264-2270.
- [7] Hayder, M. E. and Turkel, E., (1996), "On Buffer Layers as Non-Reflecting Computational Boundaries", AIAA paper 96-0273.
- [8] Hayder, M. E. and Hagstrom, T. M., (1995), "An Outflow Boundary Condition for Aeroacoustic Computations", Computational Aeroacoustics, Lyrintzis et al. edited, FED-Vol. 219, pp 41-46, ASME.
- [9] Hayder, M. E., (1995), "An Assessment of Artificial Damping Models for Aeroacoustic Calculations", AIAA paper 95-2167.

DUANE HIXON

Research Area: Aeroacoustics

The work for this year has focused on solving the linearized Euler equations for aeroacoustics applications. The solver was first validated on a test problem with a known analytic solution. Due to the unsteady nature of aeroacoustics problems, it is very important to the accuracy of the solution for the boundary conditions to allow waves to propagate correctly and not reflect from the boundaries. Several types of boundary conditions were evaluated for the test problem of a monopole in a uniform freestream [1].

The code was then applied to the problem of instability noise in a supersonic jet. Analytic and experimental data were used to validate the code for both axisymmetric and three-dimensional modes [2,3], and the code was used to perform parametric studies of the effect of heating on supersonic jet noise [4]. The code is efficient and fast-running, and the results have been very promising. Currently, parametric studies for the noise radiated by a coannular jet are being performed [5].

References

- [1] Hixon, R., Shih, S.-H., and Mankbadi, R. R., "Evaluation of Boundary Conditions for Computational Aeroacoustics", AIAA Paper 95-0160, Reno, NV, January 1995. Accepted for publication in *AIAA Journal*.
- [2] Mankbadi, R. R., Hixon, R., Shih, S.-H., and Povinelli, L. A., "On the Use of Linearized Euler Equations in the Prediction of Jet Noise", AIAA Paper 95-0505, January 1995.
- [3] Hixon, R., Shih, S.-H., and Mankbadi, R. R., "Direct Prediction of the Three-Dimensional Acoustic Field of a Supersonic Jet Using Linearized Euler Equations", CEAS/AIAA Paper 95-116, June 1995.
- [4] Hixon, R., Shih, S.-H., and Mankbadi, R. R., "Numerical Simulation of the Effect of Heating on Jet Noise", *ASME FED-Vol. 219*, 1995, p. 89-94.

- [5] Hixon, R., Shih, S.-H., and Mankbadi, R. R., "Effect of Coannular Flow on Jet Noise", abstract submitted to 2nd AIAA/CEAS Aeroacoustics Meeting, May 1996.

BO-NAN JIANG

Research Area: Flow Applications of the Least-Squares Finite Element Method

Work continued on the development and application of the least-squares finite element method (LSFEM) which is based on minimizing the residuals of the first-order system of differential equations. This is a joint research with Dr. Sheng-Tao Yu (NYMA), Dr. Jie Wu (ICOMP) and Dr. J.C. Duh (NASA Lewis, Microgravity Fluids Branch). We applied the LSFEM to the full three-dimensional computation of the surface-tension-driven convection in small square containers for which no previous numerical study has been reported. The unusual cellular patterns observed in the experiments were accurately recaptured by the numerical simulation. Work continued on two-phase flow problems. Some preliminary results have been obtained in simulating axis-symmetric swirling two-phase flow of a simplex nozzle.

KAI-HSIUNG KAO

Research Area: 3D Compressible Finite Volume Navier-Stokes Flow Solver

Prediction of the performance of a turbomachinery engine is a complex process entailing the iterative execution of aerodynamic, thermal, and structural analyses. Ultimately, engine performance is predicted from an aerodynamic analysis of the primary flowpath. However, the accuracy of the aerodynamic analysis is dependent on several factors, one of which is the definition of the flowpath geometry. Accurate determination of the flowpath geometry, which varies depending upon the operation condition, requires the inclusion of aerodynamic, thermal, and structural effects. Consequently, physical situations where heat transfer occurs between material and fluid flow with differing properties are commonly encountered in engineering practice, and often the geometries of interest are extremely irregular.

Over the last year, research work was conducted to numerically simulate the secondary flow system in a turbomachine. The goal was to investigate the coupled aero/thermal/structural problems in which the thermal response of the solid material and the flow field solution are strongly related. Although important progress has been made in solving the Navier-Stokes equations, much work is still needed to achieve robustness, accuracy, and efficiency. A newly developed flux splitting scheme, AUSM+ (Liou 1994), has been incorporated into the Navier-Stokes code. Its accuracy and efficiency are well confirmed for various flow conditions. The Chimera overset grid method and the most recently developed hybrid grid scheme, DRAGON (Kao and Liou 1994), have been successfully implemented into the code. Both grid methods show a great advantage in reducing the efforts to generate grid meshes about complex configurations. In addition, a promising feature using the turbulent subprogram developed by T.-H. Shih's Turbulence Group within ICOMP was also applied. This program which contains several most useful turbulence models including the Groups own variable C_μ model can be easily coded with the Navier-Stokes solver to perform turbulent simulation. It has been shown that the numerical results agree very well with the experimental data for supersonic compression corner problems.

A hybrid Chimera/unstructured grid method has been successfully applied for conjugate heat transfer calculations. It is designed so that the unstructured grid is used to resolve the heat conduction inside the solid body while remaining the Chimera overset grid method for the fluid flow region. The thermal condition on the solid wall requires the temperature and the heat flux values to be continuously transferred. Numerical results were compared with analytical and experimental data for a flat plate

and a C3X cooled turbine cascade. This comparison will be described in an ICOMP report and be will be presented at the ASME TURBO EXPO '96 meeting.

For future work, validation of the Navier-Stokes code will continue. Heat transfer properties which are determined by a force balance between the pressure, the centrifugal, and the buoyancy forces will be intensively studied. Using the above techniques, the entire secondary flow system within the PW4000 engine will be accurately examined and improvements for its performance will be suggested.

WILLIAM LIOU

Research Area: Turbulence Modelling

The development of advanced propulsion systems for high-speed aerospace vehicles will require accurate computational models of turbulence that can be used in the CFD calculation of individual engine component flows. If the existing models fail to deliver desirable results, new models to correct the known deficiencies must be devised in order to provide accurate predictions for flows over a wide range of, for example, engine operating conditions.

Before a new model is used in the calculation of complex flows in propulsion systems, the model has to be assessed in simple benchmark flows that not only contain the essential physical mechanisms at work in engine component flows, but also are well-documented. During this reporting period, flows were focused on which contained shock/turbulent boundary--layer interactions. The flows included supersonic ramp flows, flows with shock impingement and, transonic flows over bumps. Two new models developed at the Center for Modeling of Turbulence and Transition (CMOTT), including a high--Reynolds number and a low-Reynolds number models, were assessed for these flows. Both models were found to yield improved predictions in comparison with the existing models tested. The results of this effort were reported in technical conferences and will be submitted for publication.

Flows past multi-stage turbomachinery blades are inherently unsteady. One of the main sources of unsteadiness is due to the interaction between the blades and the viscous wakes generated by the proceeding blade rows. In the wake-induced path, unsteady, periodic boundary layers exhibit large region of transitional strips. It was shown recently that a portion of the wake-induced transitional strip relaxes to a laminar--like boundary layer in between the wake passing. Such a region is called a calmed region. A calmed region of laminar--like flow is identified in experiments as a region of decreasing but elevated wall shear behind a turbulent spot. For any transition/turbulence model to be successful in the prediction of such unsteady flows, it is essential that the dynamics of the flow is properly captured.

The second part of the research this past year, involved modeling the transitional flow past low pressure turbines.

Preliminary findings indicated that the low--Reynolds number turbulence model, which was found to perform well in the earlier studies of shock/turbulent boundary--layer interactions, is capable of mimicing certain dynamic characteristics of the flow that are similar to those found in a calmed region.

JAMES LOELLBACH

Research Area: 3D Structured Grid Generation Codes for Turbomachinery

Research focused primarily on the generation of computational grids in support of numerical flow analyses of turbomachinery components. Computational grids are required by most numerical flow

solvers to define the shapes of solid objects in the flow and to divide the space through which the fluid flows into small volumes, or cells. Certain properties of the computational grids, such as cell skewness and stretching, can strongly affect the accuracy and cost of numerical flow solutions.

During the past year, a set of programs to simplify the generation of computational grids for specific turbomachinery configurations were developed. The grid topology consists of a non-periodic H-mesh centered around a single blade in a blade row. Both axial and centrifugal machines are treated. The design goals for the programs are: 1) that they be applicable to a sufficiently wide range of configurations with a minimum of input variables, 2) that they run quickly and reliably on a wide variety of computer platforms, and 3) that they produce grids of sufficiently high quality to be used with a variety of turbomachinery flow solvers. The programs utilize an algebraic transfinite interpolation method, with additional algebraic smoothing when necessary. The required input data consist of geometry descriptions for the blade and the domain boundaries, and a small set of parameters which control the dimensions of the grid and the clustering of grid points near solid walls. Arbitrary shapes can be specified for the hub, casing, inlet and exit.

The grid generation codes have been used in support of flow analyses for a wide variety of turbomachinery problems, including transonic flows through axial and centrifugal compressors, incompressible flow through a propulsion pump, and compressible turbine flows.

In addition to the above grid generation effort, the development of an unstructured-mesh flow solver for turbomachinery flows was also pursued. It is hoped that the ability of unstructured solvers to treat geometrically complex domains will prove useful for turbomachinery analyses. This code development has been carried out primarily by Fu-Lin Tsung, also of ICOMP. During the coming year, greater participation in this project is planned. Proposed work includes modifying the flow solver to increase its applicability to general turbomachinery flows, improving the capability of generating unstructured meshes, and validating the solver by applying it to a variety of test cases.

DAVID MODIANO

Research Area: Application of Adaptive Mesh Refinement to Improve the Accuracy of Heat Transfer Calculations in Turbine Cooling Passages

In collaboration with Erlendur Steinthorsson and Philip Colella, Adaptive Mesh Refinement (AMR) was researched. The goal was to apply the AMR method to the solution of the Navier-Stokes equations on mapped grids. Previous work by Colella et al. produced an AMR algorithm for Cartesian meshes and a Godunov-based solver for the Euler equations on mapped grids. Work in this year continued the extension of the AMR algorithm to mapped grid systems. Extensive use was made of the AMR Library of C++ utility routines, developed at the Lawrence Livermore National Laboratory. The extension of the mapped grid AMR to arbitrary multiblock grid systems was nearly completed. Further work remains to be done to handle the case of three (or any quantity other than four) blocks meeting at a point. Preliminary testing of the algorithm for the viscous terms began with viscous hypersonic flows.

ANDREW T. NORRIS

Research Area: Turbulence Modelling, PDF Methods

During 1995, the NASA Lewis PDF code continued to be developed, with further model refinements and extension to 3D. In addition, work started on the creation of an automated reduced chemical mechanism code, and a NOx post-processing code.

PDF Code

Refinements of the NASA Lewis PDF code consisted of a new treatment of the boundary conditions, a new time-averaging scheme, a user controlled decoupling option and a new convection scheme. The boundary conditions were altered to include an extra row of points at the boundary. This provides a more accurate treatment of the boundaries and reduces errors in the conservation of mass. The time averaging method developed is based on a power-series weighting scheme, and requires only the storage of two sets of average values. This provides a large saving in memory required for the code. The decoupling option allows the PDF code to be run for any number of time steps, before returning to the flow solver. This feature is expected to enhance convergence of the code.

The other alteration to the code was the extension to 3D. The performance of the 3D code is expected to be tested some time in 1996, when a suitable flow solver becomes available.

Reduced Mechanisms

The development of an automated method of producing simple approximations of full chemical mechanisms was started in 1995. The method is based on the TGLDM method of Pope and Maas (1994). The basis of this approach is to identify a low-dimensional attracting manifold in scalar space, and approximate chemical reactions by movement along this manifold. The advantage of this method is that given a full chemical mechanism, a lower-order approximation can be obtained without the need of any human input. The disadvantage is the difficulty of identifying the manifold.

The TGLDM method identifies the manifold by integrating the full mechanism from cold-mix to equilibrium, with the trajectory in scalar space being an approximation to the manifold. This method has been coded and works well for simple fuels. However for more complex fuels, the integration of the full mechanism proves to be more difficult. Alternate methods involving the minimization of the Gibbs free energy are being investigated.

RICHARD PLETCHER

Research Area: Unstructured Grids

Work continued with Philip Jorgenson on the simulation of internal viscous flows using unstructured grids. A paper presented at the 1994 Aerospace Sciences Meeting (Jorgenson, P. C. E. and Pletcher R. H., "An Implicit Numerical Scheme for the Simulation of Internal Viscous Flows on Unstructured Grids," AIAA Paper 94-0306) was revised and submitted for journal publication. A more accurate means of reconstructing nodal values of variables from the cell-center values has been added to the two-dimensional unstructured code has also been enhanced to permit more accurate evaluation of wall skin friction and heat transfer. Work is also underway to incorporate turbulence modeling.

With NASA Graduate Student Researcher Tom Ramin, research was completed on the implementation of unstructured viscous solvers on parallel machines. This included the development of a three-dimensional cell-centered unstructured viscous flow code. Versions of the code were successfully run on the LACE workstation cluster at Lewis Research Center as well as the n²s at Iowa State University. A technical paper based on this research was submitted for publication. Work on unstructured grid schemes for combustion applications with NASA Graduate Student Researcher Rob Cupples continued.

In the area of low Mach number preconditioning, the method developed with Philip Jorgenson has been incorporated into two codes, an explicit Runge-Kutta scheme and an implicit LU-SGS scheme and found to work very well giving the correct behavior in the incompressible limit of the compressible equations. Work completed and underway on the large-eddy simulation of compressible flows was discussed with Lewis and ICOMP personnel.

AAMIR SHABBIR

Research Area: Turbulence Modelling

Turbomachinery flows pose a major challenge for CFD since they are three dimensional and time dependent. In order to assess the predictive capabilities of turbomachinery CFD tools a blind test case for a rotor compressor (ROTOR 37) was organized by the 1994 ASME/IGTI held at the Hague (Wisler and Denton 1994). A total of 12 groups submitted their predictions for this case, which were then compared with the detailed experimental data. The results can be found in the above reference. The predictions showed a wide scatter when compared with the experimental data. There are probably several reasons for these differences. However, one of these differences could be attributed to the deficiencies in the turbulence models but this was difficult to ascertain because different codes used the same turbulence model but gave different results. Obviously this is related to the different numerical features of the computations (type of grid, number of grid points, discretization schemes, etc.).

From the point of view of isolating the turbulence model performance, the need is to assess different turbulence models from the same numerical platform. If different turbulence models are implemented in a single CFD code and are computed on the same grid, then the differences in the results can be attributed only to the differences in the models. In this way the impact of different models on the prediction of turbomachinery flows can be assessed in a systematic manner.

With this background the present study was initiated with the objective of assessing the performance of different turbulence models for turbomachinery flows. The VSTAGE code was selected as the numerical platform to achieve this objective. ROTOR 37 flow was computed using the two equation standard $k-\epsilon$ model, and the CMOTT improved $k-\epsilon$ model. The application of the zero equation Baldwin-Lomax model has already been carried by Celestina (1994).

Detailed measurements of ROTOR 37 were carried out by Suder (1994). This included overall rotor performance as well as spanwise profiles of total pressure, total temperature, adiabatic efficiency and flow angle. The results from the computations are compared with these measurements. It is found that the CMOTT improved $k-\epsilon$ model performs both qualitatively and quantitatively better than the standard $k-\epsilon$ and the Baldwin-Lomax model. However it is noted that there are still deficiencies in predicting the experiment. Work on correcting these deficiencies is underway. (This work is being carried out in collaboration with J. Zhu and M. Celestina)

Celestina, M., 1994, "ROTOR 37 Blind Test Case", ASME/IGTI International Gas Turbine Conference, The Hague, Netherlands.

Suder, K., 1994, "ROTOR 37 Blind Test Case", ASME/IGTI International Gas Turbine Conference, The Hague, Netherlands.

Wisler, D.C. and Denton, J.D. 1994, "ROTOR 37 Blind Test Case", ASME/IGTI International Gas Turbine Conference, The Hague, Netherlands.

SHYUE-HORNG SHIH

Research Area: Aeroacoustics

Effort for developing noise prediction capabilities using computational aero-acoustics techniques was continued. Large eddy simulation of the unsteady, compressible Navier-Stokes equations was applied to a round jet to obtain the time-dependent flow and acoustics fields.

During the past year, a zonal approach for prediction of jet noise was developed. In this work, the computational domain was split into a non-linear acoustic source generation regime and a linear acoustic wave propagation regime. The unsteady flow in the non-linear acoustic source region is governed by the large-scale equations, which are the filtered compressible Navier-Stokes equations. The linearized Euler equations are used to describe the sound wave propagation in the linear acoustic region. The computed results show that no spurious wave is generated in the matching region and the computational cost is reduced by 30% when compared with the direct simulation using large-scale equations alone. This technique has been extended to the simulation of fully three-dimensional jet noise.

Second, the effect of subgrid-scale models on jet simulation was investigated. In addition to the Smagorinsky's model, the one-equation model of Yoshizawa was implemented. No substantial difference between the two solutions was observed at this moment. Further investigation is in progress, which includes the application of dynamic subgrid-scale models.

Third, the effect of multiple frequencies and random disturbance excitation on jet noise was studied. The disturbance peaks at a location closer to nozzle exit when compared to single frequency excitation. The sound pressure level contours for random disturbance excitation show two lobes of emission pattern with high frequency waves propagating at higher emission angle. The spectrum show that the dominate frequency is between Strouhal number of 0.2 and 0.4 which is consistent with experimental observations.

TSAN-HSING SHIH

Research Area: Turbulence Modelling

As in the previous year, the CMOTT research group has continually incorporated new and physically correct turbulence models into working CFD codes for propulsion systems: NPARC code, VSTAGE turbomachinery code, ALL-SPEED code, etc. Turbulence models developed at CMOTT and evaluated against various benchmark flows have also been evaluated with these working CFD codes for more complex flows. Several practical problems have been studied including NASA Lewis' Rotor 37, a transonic diffuser, a swirling can combustor, a boat-tail nozzle, a low pressure turbine, etc. Early results are very promising. These are the ongoing programs supported by both ICOMP and the project offices at NASA Lewis.

In the continuing study of turbulence modeling, the following research has been carried out: (1) surface flow in boundary layers which leads to a general wall function method for wall bounded turbulent flows; (2) a study of realizability related ODE initial value problems which leads to the implementation of realizability in transport equation models; and (3) the development of a set of models for the pressure-strain correlation term in the second moment equations. The models satisfy realizability and compare

very well with the DNS data of homogeneous turbulent shear flows. Evaluations for other flows is being pursued.

ERLENDUR STEINTHORSSON

*Research Area: Code Development for Flows in Complex Geometries such as Turbine Blade
Coolant Passages*

Research continued on the development of a general multiblock/multigrid flow solver for flows in complex geometries and on the use of adaptive mesh refinement in structured body-fitted grid systems. A new research project was also started, focusing on developing a capability to accurately simulate multi-fluid flows in (fuel-) atomizers.

Development of TRAF3D.MB: This year, the TRAF3D.MB code was modified to allow complete user control over the Runge-Kutta multi-stage scheme employed in the code to march solutions to steady state. The added control allows users to specify (a) the number of stages to use (b) the Runge-Kutta coefficients, (c) the instances when diffusion and artificial dissipation terms should be evaluated, and (d) when communication of data between blocks should take place. The code has also been equipped with a $k-\omega$ turbulence model (implemented by Dr. Ali A. Ameri, Resident Research Associate, NASA Lewis Research Center). This particular model has several desirable characteristics that make it well suited for simulations of complex internal flows. In cooperation with Dr. Ehtesham Hyder (formerly at ICOMP; currently at ICASE, NASA Langley Research Center), a separate version of the TRAF3D.MB code has been ported to a parallel computing platform. Only preliminary version of the parallel code has been prepared, further testing and modifications are planned. The TRAF3D.MB code, was used to study heat transfer in a "branced duct" geometry. The turbulence model used in that simulation was the Baldwin Lomax turbulence model. The results from the computations were found to compare admirably with available experimental data although, at specific locations in the duct, the turbulence model failed to accurately predict the effects of turbulence on the mean flow. The $k-\omega$ version of the code was applied in the study of a flow through a rectangular duct with a sharp 180 degree turn (a collaboration with Dr. David L. Rigby and Dr. Ali Ameri, Resident Research Associates, NASA Lewis Research Center). Again the computed results compared well with available experimental data.

Mesh Refinement in Body-Fitted Grid Systems (a collaboration with Professor Phillip Colella, Univ. of California, Berkeley): Last year an explicit, time-accurate flow solver for structured grid systems was developed, utilizing the AMR algorithm of Berger and Colella to enable solution--adaptive local refinement of the grid systems. This year, the code was modified to allow simulations of unsteady viscous flows. To this end, a semi-implicit scheme was utilized, treating the viscous fluxes implicitly to eliminate stability constraints that otherwise result from an explicit treatment of the viscous fluxes. Related to the ongoing work on AMR for structured grid systems, a paper was presented at a workshop on grid generation and related issues (May, 1996, NASA Lewis Research Center) advocating the use of and research on adaptive schemes for structured grid systems.

Simulations of Multi-Fluid Flows in Simplex Nozzles: (a collaboration with Dr. Kumud Ajmani, ICOMP, and Professor Gretar Tryggvason, Univ. of Michigan, Ann Arbor). A new project was started, focusing on the development of an effective and accurate methodology for simulating the formation of liquid spray in pressure swirl atomizers (simplex nozzles). The methodology pursued was originally developed by Professor Tryggvason and co-workers. The distinguishing aspect of the methodology is the use of a front-tracking algorithm to model the interface between two fluids and its motion. In the initial stage of this research, the flow is assumed to be axisymmetric to reduce computational expense

and complexity. Some preliminary results were obtained this year demonstrating the capability to simulate the formation of the conical liquid sheet that is characteristic of simplex nozzles. The research will continue into the coming year.

GERALD TRUMMER

Area: System Administration

Systems administrative tasks and software maintenance on the Aeromechanics Division's network of Silicon Graphics workstations were performed. This includes installing and upgrading graphics packages as well as making modifications to existing graphics source code as required by the needs of the scientists and researchers in the division. Work continues on integrating a CFD code developed by the researchers with graphics packages to perform real-time computation and visual analysis. Recently, movies (video tape, mpeg) of CFD work done by researchers for scientific analysis were made and displayed at remote sites by fellow researchers.

GRETAR TRYGGVASON

Research Area: Front Tracking Techniques, Simulations of Multifluid and Multiphase Solutions

During a six day visit at ICOMP last summer, work continued on methods to allow direct simulations of multi fluid and multi phase systems. The specific aspects that were worked on were the generation of efficient algorithms for topological changes of both axisymmetric and three-dimensional fronts. Those operations are essential for the modeling of the rupture of thin fluid films when two drops collide and coalesce. Also swirl was implemented into an axisymmetric version of a multi fluid code. A collaboration with Dr. Steinthorsson of ICOMP currently uses that capability to simulate the generation of fuel drops.

FU-LIN TSUNG

Research Area: Development of 3D Structured/Unstructured Hybrid Navier-Stokes Solver for Turbomachinery

The goal for the present research is to develop a reliable 3-D, Navier-Stokes, unstructured-grid solver for turbomachinery applications. In 1995, New routines have been added to allow the code to solve for rotational geometry such as rotor with tip clearance. This is accomplished by including centrifugal and Coriolis terms in the momentum and energy equations. Also, with routines developed by Dr. Frink of LaRC, the unstructured solver has been upgraded from the Runge-Kutta explicit formulation into an implicit formulation; and an one-equation Spalart-Allmaras turbulence model is now in the solver in addition to the original explicit two-equation $k-\epsilon$ turbulence model. For the inviscid flux terms, the advection upstream splitting method of Dr. M.S. Liou of NASA Lewis Research Center is now incorporated into the code. Calculations for the DLR turbine stator and Rotor37 were performed and the results were compared with other calculations and experimental data with good agreement. The solver was also used to study the flow field through a rim seal geometry, with the eventual goal of coupling the the isolated rim seal with the complete rotor and stator geometry.

ELI TURKEL

Research Area: Boundary Conditions, Oiler Solvers

Work continued on developing and comparing boundary conditions to be used at the inflow and outflow boundaries for jet acoustics problems. Most non-reflecting boundary conditions consist of some

combination of derivatives (or integrals) along a fixed outer boundary. A different idea proposed by several people is to introduce an artificial "buffer" layer at the outflow rather than just use a single line. In this buffer layer the differential equations are altered to prevent reflections. Transitions in the buffer layer must be smooth enough to prevent reflections from the gradients within the buffer layer. Two major alternatives have been developed for buffer layers. One possibility is to develop one-way equations which allow propagation of waves only away from the domain of interest with no propagation allowed back into the domain. The other alternative is to damp the waves so that their amplitude is sufficiently small that reflections are harmless. Together with Dr. M. E. Hayder (formerly of ICOMP) a project was begun to investigate an approach by Taasan to force the equations to be supersonic in the buffer layer and hence prevent back reflections. It was found that the method was useful if the buffer layer was about 1/3 the size of the physical domain so that no internal reflections occurred. A paper was presented at the AIAA meeting in Reno, 1996 with the results. Future work will also look at the Berenger approach of damping the waves in the buffer layer.

Another project is the development of preconditioning matrices for both compressible and incompressible flows. Work has been going on for a while with applications to external flow. As part of the work at ICOMP we are considering applications to internal flow and turbomachinery. Preliminary tests have indicated fast convergence for a two dimensional internal flow coupled with multigrid. Future work will continue applications to three dimensional stator-rotor configurations. One of the theoretical aspects is the different boundary conditions needed at the inflow.

JIE WU

Research Area: Development of Finite Element Methods for Fluid Mechanics and Electromagnetics

During the year 1995 research work was performed in several aspects of computational mechanics. This has been a continuation of the collaborative research work on the least-squares finite element method (LSFEM) with Dr. Bo-Nan Jiang (ICOMP) and Dr. Sheng-Tao Yu (NYMA Setar Team). Ultimately this will lead to a general purpose, multi-disciplinary computer analysis package for engineering problems related to propulsion systems.

1. Implementation of turbulence models

The standard $k-\epsilon$ turbulence model (collaborative work with the CMOTT group at ICOMP) was incorporated into the least-squares finite element method (LSFEM) code. Both the low Reynolds number model and the wall function approach have been implemented and tested on a flat plate boundary layer problem. The introduction of turbulence calculations further enhances the capability of the LSFEM flow solver. The entire computer code was written in a highly modularized manner such that the calculation of both the mean flow and the turbulent quantities are almost independent of each other, and the two parts interact with each other through the turbulent viscosity. This allows great flexibility for further testing and development. For example, the present package can easily be modified to combine the LSFEM mean flow solver with the turbulent viscosity provided through other numerical approaches; and the LSFEM turbulence solver can also receive mean flow quantities input from other solvers and feed back the turbulent viscosity and other turbulent quantities to the mean flow solver.

2. Simulating free surface flows

This procedure is based on a continuum approach first developed by Brackbill, et al. In this approach fluids of different properties are identified through the use of a color function. The interfaces between

different fluids are represented as transition regions with finite thicknesses.

The interfaces are distinct due to sharp gradients of the color function. The evolution of the interfaces is captured by solving the convective equation of the color function. The surface tension effect is presented in terms of stresses. The stresses are defined by the gradient of the color function. In its analytical form, this stress formulation is equivalent to the original continuum surface force (CSF) model. Numerically, however, the use of the stress formulation has some advantages over the original CSF model, as it bypasses the difficulty in approximating the curvature of the interfaces.

The LSFEM is used to discretize the governing equation systems. The LSFEM handles all the equations in a unified manner without any additional special treatment such as upwinding or artificial dissipation. Various bench mark tests have been carried out for two-dimensional planar and axis-symmetric flows. Some preliminary results have also been obtained in simulating axis-symmetric two-phase swirling flow of a simplex nozzle.

3. Further work on parallel computation

The parallelization of the LSFEM general purpose partial differential equation solver on the distributed memory systems composed of IBM work station clusters, i.e. the LACE (Lewis Advanced Cluster Environment) system was advanced further during FY95.

4. Some initial effort to speed up the current LSFEM solver

Some basic multigrid iteration strategy was introduced into the general purpose LSFEM solver. Preliminary tests reveal a definite speedup for nearly linear systems (e.g. highly viscous flows). For highly non-linear systems no obvious speedup has been observed and further investigation is necessary before any conclusion can be reached.

ZHIGANG YANG

Research Area: Turbulence Modelling

In the past year, research activity has been focused on the application of two-equation turbulence models to turbulent and transitional flows in propulsion systems. This work involved three projects.

1. Turbulence model development for NPARC (with Drs. J. Zhu and T.H. Shih of ICOMP) --- In applying turbulence models to flows within propulsion systems, the NPARC code was chosen as the base code. NPARC is a general purpose code widely used in the propulsion systems community. The overall mission in this project was to enhance the turbulence modeling capability. The work has been incorporated into the development plan of the NPARC Alliance, a partnership between NASA Lewis Research Center and the Arnold Engineering Development Center. In order to create a user-friendly environment for NPARC's turbulence capabilities, a separate turbulence subprogram was constructed by Dr. Zhu. Currently, a number of two-equation turbulence models are available in the subprogram, including an improved model developed within CMOTT. A major feature of the CMOTT model is that it is realizable, i.e., no physically unrealistic Reynolds stresses will be produced for any mean field conditions. This feature allows a robust calculation of complex turbulent flows in propulsion systems. Models in the subprogram have been tested against typical flows found in propulsion systems, including inlet flows, nozzle flows, and diffuser flows. Results from these computations were reported in the

1995 AIAA Joint Propulsion Conference, and the 1996 AIAA Aerospace Sciences Meeting, respectively.

2. Modeling of turbulent/transitional flows in LPT (with Drs. W. Liou and T.H. Shih of ICOMP) -- The Low Pressure Turbine (LPT) is an important component in aero-propulsion engines. Bypass transition plays a crucial role in flows over a LPT, due to the relatively low flow Reynolds numbers, the high level of disturbances in the environment, and the passage of wakes. The mission of this research work is to provide the engine industry with a computational model for LPT flow analysis. This research project is a part of NASA's Low Pressure Turbine Flow Physics Program. Preliminary calculations suggest that the CMOTT two-equation model has the capability to mimic the 'calmed region', an important phenomenon that has been experimentally observed in LPT flows. The results were presented in the 1995 LPT Flow Physics Workshop.

3. Turbulence modeling capability for LSFEM (with Drs. J. Wu, B.N. Jiang, and T.H. Shih of ICOMP) -- Least Square Finite Element Method (LSFEM) is a novel approach to the analysis and computation of fluid flows. It has a solid theoretical basis for analysis and the flexibility to handle complex geometries in practical computations. LSFEM differs from the traditional Galerkin-type finite element method in that it can handle problems when the convection term is important, an important feature of fluid flow problems. Many laminar flows have been calculated using this method by Dr. Jiang and his co-workers, yielding impressive results. The purpose of this project is to extend this method to turbulent flows. As a first step, turbulent channel flows and turbulent boundary layer flows have been successfully calculated for the purpose of code validation.

SHAYE YUNGSTER

Research Area: High Speed Combustion and Detonation Waves

The research work for the 1995 year focused on three different areas:

- 1) Implementation of the chimera grid scheme into the ALLSPD-3D code.

The objective of this work is to implement the chimera grid scheme into the ALLSPD-3D code, and to evaluate the effectiveness of the combined version in the computation of combustor flowfields. The chimera grid scheme has not been applied to reacting flows previously.

The ALLSPD-3D code is being developed at the NASA Lewis Research Center and the University of Toledo for efficient solution of combustor flowfields at all speeds. This code solves the three-dimensional chemical non-equilibrium Navier-Stokes equations over a wide range of Mach numbers.

In addition to having efficient numerics and accurate physical models, the ALLSPD-3D code must be able to handle complex geometries. An effective approach for this purpose is the domain decomposition method known as the chimera scheme. This approach uses a composite of independent overlapping grids to discretize a complex geometrical configuration. The main advantage of the chimera method is a simplification of the grid generation process by permitting a modular approach.

The chimera grid scheme has been successfully incorporated into the ALLSPD-3D code. The combined ALLSPD-3D/chimera grid approach has been applied to several problems such as channel flow, steady and unsteady flow over a bluff-body, bluff-body stabilized spray and premixed combustion flames, turbomachinery flows and AST combustion test rig. Additional benchmark computations are currently

being carried out. The results of this work were presented at the 34th Aerospace Sciences Meeting, AIAA paper 96-0731 (in collaboration with K.-H. Chen of the University of Toledo).

2) CFD Analysis of Rocket-Based Combined Cycle (RBCC) propulsion system

Background

The RBCC engine has the potential to offer better performance than a pure rocket for high speed cruise applications, highly-reusable launch vehicles, and accordingly the NASA X-plane initiative. Analytical and experimental work aimed at evaluating the RBCC concept is being conducted by a team consisting of NASA, USAF, and various contractors. Sea-level static and high Mach number freejet tests of this concept will be carried out at the NASA Plumbrook Hypersonic Tunnel Facility (HTF).

The objective of the present work is to develop predictive CFD tools in support of the experimental effort. In particular, an inlet analysis is being provided, help is being provided to determine operability limits, and help is being provided to optimize the engine geometry. In addition, an analysis of the combustor and fuel injection process will be completed soon.

Approach

Currently the NPARC code is being used for the inlet and overall flowfield analysis. This code is a fully three-dimensional Navier-Stokes solver, with capabilities for handling complex geometries. It can be used to simulate steady-state or transient flows in a very efficient manner.

The flowfield for two engine configurations at various Mach numbers has been computed and the inlet flow starting characteristics have been studied. Current calculations are aimed at simulating experiments with back-pressured engine conditions. For the combustor analysis, the plan is to use either the RPLUS or ALLSPD-3D codes. Results of this work will be presented at the 32nd Joint Propulsion Conference.

3) CFD Studies of flow establishment in hypersonic facilities

Background

A major difficulty associated with ground testing of hypersonic propulsion systems in pulse facilities is the short test time available (on the order of a millisecond). In some cases, the test time may be less than the time required to fully establish the reacting flow, especially if recirculation zones and shock wave/boundary layer or detonation wave/boundary layer interactions are present. Numerical simulation of the temporal evolution of the combustion process can supplement experimental work by providing detailed information about reaction initiation and flow establishment time in pulse facilities.

Approach

A computational fluid dynamics code that addressed the need for efficient time-accurate simulations of chemically reacting, viscous flows has been developed. It is based on a spatially second order total variation diminishing (TVD) scheme and a temporally second order, implicit, variable-step, backward differentiation formula (BDF) method. The inversion of large matrices is avoided by partitioning the system into reacting and nonreacting parts; a fully coupled interaction is, nonetheless, maintained.

The new code has been applied to the investigation of the combustion process generated by the interaction between an axisymmetric projectile and an explosive gas mixture that has been accelerated to hypersonic speeds in an expansion tube. Also, the temporal evolution of the shock-induced combustion process in a ram accelerator has been investigated. In particular, the transition from the launch tube into the ram accelerator section containing an explosive hydrogen-oxygen-argon gas mixture was studied. The hydrogen-oxygen chemistry was modeled with a 9-species, 19-reaction mechanism.

The results of this work are presented in two ICOMP reports (ICOMP-95-19, ICOMP-95-20) and in articles to appear in the Shock Waves Journal.

JIANG ZHU

Research Area: Turbulence Modelling

Research in 1995 mainly focused on the development and validation of two turbulence modules, one for the NPARC code used extensively by the U.S. propulsion community, and the other for the VSTAGE code developed at the Lewis Center to simulate turbomachinery flows.

In the NPARC module, three low Reynolds number two equation turbulence models have been implemented: Chien, Shih-Lumley and CMOTT realizable models. The module has been applied to calculate a number of flows including flows over a flat plate, in an ejector nozzle, in a transonic diffuser, and a boat-tail nozzle flow. For all the flow cases tested so far, it has been found that both the Shih-Lumley and CMOTT models produce improved or similar predictions compared with the Chien model, and the CMOTT model turns out to be more computationally robust than the other two. It was able to give numerical solutions in cases when the others suffered from numerical instability. The details of the module and its applications were reported in the two AIAA papers: AIAA 95-2612 and 95-2761, both of which were presented at the 31st Joint Propulsion Conference, July 1995. The compressible wall functions based on the Van Driest transformation has also been implemented into the two dimensional version of the NPARC module. Code validation and the extension to a three-dimensional version are under way.

The VSTAGE module has the same numerical framework as that of the NPARC module, except the following unique features: (a) a cylindrical coordinate system is used as the absolute (fixed) reference frame, and correspondingly the cylindrical coordinate components of the flow velocity are used in the curvilinear transformation of equations; (b) rotational effects are taken into account by transforming the absolute reference frame to a relative (rotating) frame; (c) instead of the commonly used metrics, the module uses volumes and areas of control volumes as the major geometric quantities, which is in line with the definition of variables in VSTAGE; (d) the data transfer between VSTAGE and the module is via the Fortran common blocks; and (e) three high Reynolds number two equation turbulence models have been incorporated, one standard and two CMOTT realizable models. The CMOTT models differ from each other in the dissipation rate equation, one using the standard equation and the other the newly developed equation. The wall function approach is used to formulate the solid wall boundary conditions. The module were validated by calculating the ROTOR 37 flow which was a blind test case for 1994 ASME/IGTI International Gas Turbine Conference. It has been found that the prediction of the VSTAGE code equipped with the module is much better than all the previous predictions presented at the conference.

Reports and Abstracts

1995 REPORTS AND ABSTRACTS

Feiler, Charles E., Compiler and Editor: "Institute for Computational Mechanics in Propulsion (ICOMP), Ninth Annual Report - 1994", ICOMP Report 95-1, NASA TM 106884, March, 1995, 53 pages.

The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by the Ohio Aerospace Institute (OAI) and funded under a cooperative agreement by the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities and accomplishments during 1994.

Coquel, Frédéric (ICOMP); and Liou, Meng-Sing (NASA Lewis): "Hybrid Upwind Splitting (HUS) by a Field-by-Field Decomposition", ICOMP Report 95-2, NASA TM 106843, January, 1995, 91 pages.

We introduce and develop in this paper a new approach for upwind biasing: the *Hybrid Upwind Splitting* (HUS) method. This original procedure is based on a suitable hybridization of current prominent *Flux Vector Splitting* (FVS) and *Flux Difference Splitting* (FDS) methods. The HUS method is designed to naturally combine the respective strengths of the above methods while excluding their main deficiencies. Specifically, the HUS strategy yields a family of upwind methods that exhibit the *robustness of FVS schemes in the capture of nonlinear waves* and the *accuracy of some FDS schemes in the resolution of linear waves*. We give a detailed construction of the HUS methods following a *general and systematic procedure* directly performed at the basic level of the *Field by Field* (i.e. waves) decomposition involved in FDS methods. For such a given decomposition, each field is endowed either with FVS or FDS numerical fluxes, depending on the nonlinear nature of the field under consideration. Such a design principle is made possible thanks to the introduction of a convenient formalism that provides us with a unified framework for upwind methods. The HUS methods we propose bring significant improvements over current methods in terms of accuracy and robustness. They yield *entropy-satisfying* approximate solutions as they are strongly supported in numerical experiments. Field by field hybrid numerical fluxes also achieve *fairly simple and explicit* expressions and hence require a computational effort between that of the FVS and FDS. Several numerical experiments ranging from stiff 1D shock-tube to high speed viscous flows problems are displayed, intending to illustrate the benefits of the present approach. We shall assess in particular the relevance of our HUS schemes to viscous flow calculations.

Thakur, Siddharth (University of Florida); Shyy, Wei (ICOMP); and Liou, Meng-Sing (NASA Lewis): "Investigation of Convection and Pressure Treatment with Splitting Techniques", ICOMP Report 95-3, NASA TM 106868, March, 1995, 39 pages.

Treatment of convective and pressure fluxes in the Euler and Navier-Stokes equations using splitting formulas for convective velocity and pressure is investigated. Two-schemes -- Controlled Variation Scheme (CVS) and Advection Upstream Splitting Method (AUSM) -- are explored for their accuracy in resolving sharp gradients in flows involving moving or reflecting shock waves as well as a one-dimensional combusting flow with a strong heat release source term. For two-dimensional compressible flow computations, these two schemes are implemented in one of the pressure-based algorithms, whose very basis is the separate treatment of convective and pressure fluxes. For the convective fluxes in the momentum equations as well as

the estimation of mass fluxes in the pressure correction equation (which is derived from the momentum equations and continuity equations) of the present algorithm, both first- and second order (with minmod limiter) flux estimations are employed. Some issues resulting from the conventional use in pressure-based methods of a staggered grid, for the location of velocity components and pressure, are also addressed. Using the second-order fluxes, both CVS and AUSM type schemes exhibit sharp resolution. Overall, the combination of upwinding and splitting for the convective and pressure fluxes separately exhibits robust performance for a variety of flows and is particularly amenable for adoption in pressure-based methods.

Liu, J. (University of Florida); and Shyy, W. (ICOMP): "On the Conservative Interface Treatment for Multi-Block Viscous Flow Computations", ICOMP Report 95-4, NASA CR 195442, March, 1995, 35 pages.

A pressure-based multi-block computational method is developed for solving the incompressible Navier-Stokes equations in general curvilinear grid systems. The scheme is based on the semi-implicit type flow solver with the staggered grid. Issues concerning the mass and momentum flux treatments at the discontinuous grid interface are addressed. Systematic numerical experiments for different interface treatments involving (i) straightforward interpolation, (ii) globally conservative scheme, and (iii) locally conservative scheme have been conducted. It is demonstrated that mass conservation has to be maintained locally, at the grid interface, with accuracy compatible with that of the scheme used in interior domain. Direct interpolation or globally conservative interface treatment of mass flux can not yield solutions with desirable accuracy.

Shih, S. H., Hixon, R. (ICOMP); and Mankbadi, Reda R. (NASA Lewis): "Three-Dimensional Structure in a Supersonic Jet: Behavior Near Centerline", ICOMP Report 95-5, NASA TM 106869, AIAA-95-0681, January, 1995, 13 pages.

The unsteady structure of a supersonic jet is highly three dimensional, though the mean flow is axisymmetric. In simulating a circular jet, the centerline represents a computational boundary. As such, spurious modes can be generated near centerline, unless special attention is given to the behavior of the 3D structure near the centerline. Improper treatment of the dependent variables near the centerline results in the solution diverging or being suitable only for small amplitude excitation. With a careful treatment of the centerline formulation, no spurious mode is generated. The results show that a near linear disturbance growth is obtained, as the linear stability theory indicates. At high levels of excitation, nonlinear development of disturbances is evident and saturation is reached downstream.

Mankbadi, R. R. (NASA Lewis); Shih, S. H., Hixon, R. (ICOMP); and Povinelli, L. A. (NASA Lewis): "Direct Computation of Sound Radiation by Jet Flow Using Large-Scale Equations", ICOMP Report 95-6, NASA TM 106877, AIAA-95-0680, January, 1995, 12 pages.

Jet noise is directly predicted using large-scale equations. The computational domain is extended in order to directly capture the radiated field. As in conventional large-eddy-simulations, the effect of the unresolved scales on the resolved ones is accounted for. Special attention is given to boundary treatment to avoid spurious modes that can render the computed fluctuations totally unacceptable. Results are presented for a supersonic jet a Mach number 2.1.

Gajjar, J. S. B. (ICOMP): "Nonlinear Stability of Non-Stationary Cross-Flow Vortices in Compressible Boundary Layers", ICOMP Report 95-7, NASA CR 189184, May, 1995, 34 pages.

The nonlinear evolution of long wavelength non-stationary cross-flow vortices in a compressible boundary layer is investigated and the work extends that of Gajjar (1994) to flows involving multiple critical layers. The basic flow profile considered in this paper is that appropriate for a fully three-dimensional boundary layer with $O(1)$ Mach number and with wall heating or cooling. The governing equations for the evolution of the cross-flow vortex are obtained and some special cases are discussed. One special case includes linear theory where exact analytic expressions for the growth rate of the vortices are obtained. Another special case is a generalization of the Bassom & Gajjar (1988) results for neutral waves to compressible flows. The viscous correction to the growth rate is derived and it is shown how the unsteady nonlinear critical layer structure merges with that for a Haberman type of viscous critical layer.

Jiang, Bo-nan, Wu, Jie (ICOMP); and Povinelli, L. A. (NASA Lewis): "The Origin of Spurious Solutions in Computational Electromagnetics", ICOMP Report 95-8, NASA TM 106921, May, 1995, 46 pages.

The origin of spurious solutions in computational electromagnetics, which violate the divergence equations, is deeply rooted in a misconception about the first-order Maxwell's equations and in an incorrect derivation and use of the curl-curl equations. The divergence equations must be always included in the first-order Maxwell's equations to maintain the ellipticity of the system in the space domain and to guarantee the uniqueness of the solution and/or the accuracy of the numerical solutions. The div-curl method and the least-squares method provide rigorous derivation of the equivalent second-order Maxwell's equations and their boundary conditions. The node-based least squares finite element method (LSFEM) is recommended for solving the first-order full Maxwell equations directly. Examples of the numerical solutions by LSFEM for time-harmonic problems are given to demonstrate that the LSFEM is free of spurious solutions.

Mankbadi, Reda R. (NASA Lewis); Hixon, R. And S.-H. Shih (ICOMP); and Povinelli, Louis A. (NASA Lewis): "On the Use of Linearized Euler Equations in the Prediction of Jet Noise", ICOMP Report 95-9, NASA TM 106929, AIAA 95-0505, May, 1995, 14 pages.

Linearized Euler equations are used to simulate supersonic jet noise generation and propagation. Special attention is given to boundary treatment. The resulting solution is stable and nearly free from boundary reflections without the need for artificial dissipation, filtering, or a sponge layer. The computed solution is in good agreement with theory and observation and is much less CPU-intensive as compared to large-eddy simulations.

Hayder, M. Ehtesham (ICOMP); and Hagstrom, Thomas (ICOMP): "An Outflow Boundary Condition for Aeroacoustic Computations", ICOMP Report 95-10, NASA TM 106930, May, 1995, 8 pages.

We present a formulation of boundary condition for flows with small disturbances. We test our methodology in an axisymmetric jet flow calculation, using both the Navier-Stokes and Euler equations. Solutions in the far field are assumed to be oscillatory. If the oscillatory disturbances are small, the growth of the solution variables can be predicted by linear theory. We use the eigenfunctions of the linear theory explicitly in our formulation of the boundary conditions. This guarantees correct solutions at the boundary in the limit where the predictions of linear theory are valid.

Hayder, M. Ehtesham (ICOMP): "An Assessment of Artificial Damping Models for Aeroacoustic Calculations", ICOMP Report 95-11, NASA CR 198354, AIAA 95-2167, June, 1995, 18 pages.

We present a study of the effect of artificial dissipation models on nonlinear wave computations using a few high order schemes. Our motivation is to assess the effectiveness of artificial dissipation models for their suitability for aeroacoustic computations. We solve three model problems in one dimension using the Euler equations. Initial conditions are chosen to generate nonlinear waves in the computational domain. We examine various dissipation models in central difference schemes such as the Dispersion Relation Preserving (DRP) scheme and the standard fourth and sixth order schemes. We also make a similar study with the fourth order MacCormack scheme due to Gottlieb and Turkel.

Zhu, J. And Shih, T.-H. (ICOMP): "A Turbulence Module for the NPARC Code", ICOMP Report 95-12, NASA CR 198358, AIAA 95-2612, June, 1995, 12 pages.

A turbulence module is developed for the 2D version of the NPARC code which is currently restricted to planar or axisymmetric flows without swirling. Four turbulence models have been built into the module: Baldwin-Lomax, Chien, Shih-Lumley and CMOTT models. The first is a mixing-length eddy-viscosity model which is mainly used for initialization of computational fields and the last three are the low Reynolds number two-equation models. Unlike Chien's model, both the Shih-Lumley and CMOTT models do not involve the dimensionless wall distance y^+ , an advantage for separated flow calculations. Contrary to the NPARC and most other compressible codes, the non-delta form of transport equations is used which leads to a simpler linearization and is more effective than using the delta form in ensuring the positiveness of the turbulent kinetic energy and its dissipation rate. To reduce numerical diffusion while maintaining necessary stability, a second-order accurate and bounded scheme is used for the convective terms of the turbulent transport equations. This scheme is implemented in a deferred correction manner so that the main coefficients of the resulting difference equations are always positive, thus making the numerical solution process unconditionally stable. The system of equations are solved via a decoupled method and by the alternating direction TDMA of Thomas. The module can be easily linked to the NPARC code for turbulent flow calculations.

Shih, Tsan-Hsing (ICOMP); and Lumley, John L. (Cornell University): "Applications of Direct Numerical Simulation of Turbulence in Second Order Closures", ICOMP Report 95-13, CMOTT 95-2, NASA CR 198386, September, 1995, 20 pages.

This paper discusses two methods of developing models for the rapid pressure-strain correlation term in the Reynolds stress transport equation using direct numerical simulation (DNS) data. One is perturbation about isotropic turbulence, the other is perturbation about two-component turbulence-an extremely anisotropic turbulence. A model based on the latter method is proposed and is found to be very promising when compared with DNS data and other models.

Messiter, A. F. (ICOMP): "Large-Amplitude Long-Wave Instability of a Supersonic Shear Layer", ICOMP Report 95-14, NASA CR 198401, September, 1995, 44 pages.

For sufficiently high Mach numbers, small disturbances on a supersonic vortex sheet are known to grow in amplitude because of slow nonlinear wave steepening. Under the same external conditions, linear theory predicts slow growth of long-wave disturbances to a thin supersonic shear layer. An asymptotic formulation is given here which adds nonzero shear-layer thickness

to the weakly nonlinear formulation for a vortex sheet. Spatial evolution is considered, for a spatially periodic disturbance having amplitude of the same order, in Reynolds number, as the shear-layer thickness. A quasi-equilibrium inviscid nonlinear critical layer is found, with effects of diffusion and slow growth appearing through a nonsecularity condition. Other limiting cases are also considered, in an attempt to determine a relationship between the vortex-sheet limit and the long-wave limit for a thin shear layer; there appear to be three special limits, corresponding to disturbances of different amplitudes at different locations among the shear layer.

Hagstrom, Thomas (ICOMP): "On High-Order Radiation Boundary Conditions", ICOMP Report 95-15, NASA CR 198404, September, 1995, 23 pages.

In this paper we develop the theory of high-order radiation boundary conditions for wave propagation problems. In particular, we study the convergence of sequences of time-local approximate conditions to the exact boundary condition, and subsequently estimate the error in the solutions obtained using these approximations. We show that for finite times the Padé approximants proposed by Engquist and Majda lead to exponential convergence if the solution is smooth, but that good long-time error estimates cannot hold for spatially local conditions. Applications in fluid dynamics are also discussed.

Gajjar, J.S.B. (ICOMP): "On the Nonlinear Evolution of a Stationary Cross-Flow Vortex in a Fully Three-Dimensional Boundary Layer Flow", ICOMP Report 95-16, NASA CR 198405, October, 1995, 38 pages.

We consider the nonlinear stability of a fully three-dimensional boundary layer flow in an incompressible fluid and derive an equation governing the nonlinear development of a stationary cross-flow vortex. The amplitude equation is a novel integro-differential equation which has spatial derivatives of the amplitude occurring in the kernel function. It is shown that the evolution of the cross-flow vortex is strongly coupled to the properties of an unsteady wall layer which is in fact driven by an unknown slip velocity, proportional to the amplitude of the cross-flow vortex. The work is extended to obtain the corresponding equation for rotating disk flow. A number of special cases are examined and the numerical solution for one of the cases, and further analysis, demonstrates the existence of finite-distance as well as focussing type singularities. The numerical solutions also indicate the presence of a new type of nonlinear wave solution for a certain set of parameter values.

Center for Modeling of Turbulence and Transition (CMOTT): "Center for Modeling of Turbulence and Transition: Research Briefs--1995", ICOMP Report 95-17, CMOTT 95-3, NASA CR 198408, October, 1995, 114 pages.

This research brief contains the progress reports of the research staff of the Center for Modeling of Turbulence and Transition (CMOTT) from July 1993 to July 1995. It also constitutes a progress report to the Institute of Computational Mechanics in Propulsion located at Ohio Aerospace Institute and the Lewis Research Center. CMOTT has been in existence for about four years. In the first three years, its main activities were to develop and validate turbulence and combustion models for propulsion systems, in an effort to remove the deficiencies of existing models. Three workshops on computational turbulence modeling were held at LeRC (1991, 1993, 1994). At present, CMOTT is integrating the CMOTT developed/improved models into CFD tools which can be used by the propulsion systems community. This activity has resulted in an increased collaboration with the Lewis CFD researchers.

Shih, S.H.; Hixon, D.R. (ICOMP); and Mankbadi, Reda (NASA Lewis): "A Zonal Approach for Prediction of Jet Noise", ICOMP Report 95-18, CEAS/AIAA-95-144, NASA TM 107075, June, 1995, 12 pages.

A zonal approach for direct computation of sound generation and propagation from a supersonic jet is investigated. The present work splits the computational domain into a non-linear, acoustic-source regime and linear acoustic wave propagation regime. In the non-linear regime, the unsteady flow is governed by the large-scale equations, which are the filtered compressible Navier-Stokes equations. In the linear acoustic regime, the sound wave propagation is described by the linearized Euler equations. Computational results are presented for a supersonic jet at $M=2.1$. It is demonstrated that no spurious modes are generated in the matching region and the computational expense is reduced substantially as opposed to fully large-scale simulation.

Yungster, S. (ICOMP); and Radhakrishnan, K. (NYMA, Inc.): "Computational Study of Flow Establishment in Hypersonic Pulse Facilities", ICOMP Report 95-19, NASA CR 198407, November, 1995, 16 pages.

This paper presents a study of the temporal evolution of the combustion flowfield established by the interaction of ram-accelerator-type projectiles with an explosive gas mixture accelerated to hypersonic speeds in an expansion tube. The Navier-Stokes equations for a chemically reacting gas are solved in a fully coupled manner using an implicit, time accurate algorithm. The solution procedure is based on a spatially second order, total variation diminishing (TVD) scheme and a temporally second order, variable-step, backward differentiation formula method. The hydrogen-oxygen chemistry is modeled with a 9-species, 19-step mechanism. The accuracy of the solution method is first demonstrated by several benchmark calculations. Numerical simulations of expansion tube flowfields are then presented for two different configurations. In particular, the development of the shock-induced combustion process is followed. In one case, designed to ensure ignition only in the boundary layer, the lateral extent of the combustion front during the initial transient phase was surprisingly large. The time histories of the calculated thrust and drag forces on the ram accelerator projectile are also presented.

Yungster, Shaye (ICOMP); Radhakrishnan, K. (NYMA, Inc.); and Rabinowitz, M.J. (NASA Lewis): "Computational Study of Flow Establishment in a Ram Accelerator", ICOMP Report 95-20, AIAA-95-2489, NASA TM 107068, July, 1995, 13 pages.

The temporal evolution of the combustion process established during projectile transition from the launch tube into the ram accelerator section containing an explosive hydrogen-oxygen-argon gas mixture is studied. The Navier-Stokes equations for chemically reacting flow are solved in a fully coupled manner, using an implicit, time accurate algorithm. The solution procedure is based on a spatially second order total variation diminishing scheme and a temporally second order, variable-step, backward differentiation formula method. The hydrogen-oxygen chemistry is modeled with a 9-species, 19-step mechanism. The accuracy of the solution method is first demonstrated by several benchmark calculations. Numerical simulations of two ram accelerator configurations are then presented. In particular, the temporal developments of shock-induced combustion and thrust forces are followed. Positive thrust is established in both cases; however, in one of the ram accelerator configurations studied, combustion in the boundary layer enhances its separation, ultimately resulting in unstart.

Zhu, J.; Shih, T.-H. (ICOMP): "Calculations of Diffuser Flows With an Anisotropic K- ϵ Model", ICOMP Report 95-21, CMOTT-95-4, NASA CR 198418, November, 1995, 12 pages.

A newly developed anisotropic K- ϵ model is applied to calculate three axisymmetric diffuser flows with or without separation. The new model uses a quadratic stress-strain relation and satisfies the realizability conditions, i.e., it ensures both the positivity of the turbulent normal stresses and the Schwarz' inequality between any fluctuating velocities. Calculations are carried out with a finite-volume method. A second-order accurate, bounded convection scheme and sufficiently fine grids are used to ensure numerical credibility of the solutions. The standard K- ϵ model is also used in order to highlight the performance of the new model. Comparison with the experimental data shows that the anisotropic K- ϵ model performs consistently better than does the standard K- ϵ model in all of the three test cases.

Steinthorsson, Erlendur; Modiano, David (ICOMP); Colella, Phillip (University of California): "Adaptive Mesh Refinement in Curvilinear Body-Fitted Grid Systems", ICOMP Report 95-22, Reprint from CP-3316, November, 1994, 8 pages.

To be truly compatible with structured grids, an AMR algorithm should employ a block structure for the refined grids to allow flow solvers to take advantage of the strengths of structured grid systems, such as efficient solution algorithms for implicit discretizations and multigrid schemes. One such algorithm, the AMR algorithm of Berger and Colella, has been applied to an adapted for use with body-fitted structured grid systems. Results are presented for a transonic flow over a NACA0012 airfoil (AGARD-03 test case) and a reflection of a shock over a double wedge.

Steinthorsson, Erlendur (ICOMP); and Ameri, Ali A. (AYT Corporation): "Computations of Viscous Flows in Complex Geometries Using Multiblock Grid Systems", ICOMP Report 95-23, AIAA-95-0177, NASA CR, January, 1995, 12 pages.

Generating high quality, structured, continuous, body-fitted grid systems (multiblock grid systems) for complicated geometries has long been a most labor-intensive and frustrating part of simulating flows in complicated geometries. Recently, new methodologies and software have emerged that greatly reduce the human effort required to generate high quality multiblock grid systems for complicated geometries. These methods and software require minimal input from the user-typically, only information about the topology of the block structure and number of grid points. This paper demonstrates the use of the new breed of multiblock grid systems in simulations of internal flows in complicated geometries. The geometry used in this study is a duct with a sudden expansion, a partition and an array of cylindrical pins. This geometry has many of the features typical of internal coolant passages in turbine blades. The grid system used in this study was generated using a commercially available grid generator. The simulations were done using a recently developed flow solver, TRAF3D.MB, that was specially designed to use multiblock grid systems.

Steinthorsson, Erlendur; Modiano, David (ICOMP); Crutchfield, William; Bell, John B. (Lawrence Livermore National Laboratory); and Colella, Phillip (University of California): "An Adaptive Semi-Implicit Scheme for Simulations of Unsteady Viscous Compressible Flows", ICOMP Report 95-24, AIAA-95-1727, NASA CR 198420, June, 1995, 13 pages.

A numerical scheme for simulation of unsteady, viscous, compressible flows is considered. The scheme employs an explicit discretization of the inviscid terms of the Navier-Stokes equations and an implicit discretization of the viscous terms. The discretization is second order accurate

in both space and time. Under appropriate assumptions, the implicit system of equations can be decoupled into two linear systems of reduced rank. These are solved efficiently using a Gauss-Seidel method with multigrid convergence acceleration. When coupled with a solution-adaptive mesh refinement technique, the hybrid explicit-implicit scheme provides an effective methodology for accurate simulations of unsteady viscous flows. The methodology is demonstrated for both body-fitted structured grids and for rectangular (Cartesian) grids.

Hariharan, S.I. (ICOMP); Stenger, Edward J. (University of Akron); Scott, J.R.(NASA Lewis): "Potential Theoretic Methods for Far Field Sound Radiation Calculations", ICOMP Report 95-25, NASA TM 107118, December, 1995, 32 pages.

In the area of computational acoustics, procedures which accurately predict the far-field sound radiation are much sought after. A systematic development of such procedures are found in a sequence of papers by Atassi. The method presented here is an alternate approach to predicting far field sound based on simple layer potential theoretic methods. The main advantages of this method are: it requires only a simple free space Green's function, it can accommodate arbitrary shapes of Kirchhoff surfaces, and is readily extendable to three-dimensional problems. Moreover, the procedure presented here, though tested for unsteady lifting airfoil problems, can easily be adapted to other areas of interest, such as jet noise radiation problems. Results are presented for lifting airfoil problems and comparisons are made with the results reported by Atassi. Direct comparisons are also made for the flat plate case.

Hagstrom, T.; Hariharan, S.I. (ICOMP): "Progressive Wave Expansions and Open Boundary Problems", ICOMP Report 95-26, NASA CR 198432, December, 1995, 24 pages.

In this paper we construct progressive wave expansions and asymptotic boundary conditions for wave-like equations in exterior domains, including applications to electromagnetics, compressible flows and aeroacoustics. The development of the conditions will be discussed in two parts. The first part will include derivations of asymptotic conditions based on the well-known progressive wave expansions for the two-dimensional wave equations. A key feature in the derivations is that the resulting family of boundary conditions involve a single derivative in the direction normal to open boundary. These conditions are easy to implement and an application in electromagnetics will be presented. The second part of the paper will discuss the theory for hyperbolic systems in two dimensions. Here, the focus will be to obtain the expansions in a general way and to use them to derive a class of boundary conditions that involve only time derivatives or time and tangential derivatives. Maxwell's equations and the compressible Euler equations are used as examples. Simulations with the linearized Euler equations are presented to validate the theory.

Kao, Kai-Hsiung (ICOMP); Liou, Meng-Sing (NASA Lewis): "On the Application of Chimera/Unstructured Hybrid Grids for Conjugate Heat Transfer", ICOMP Report 95-27, NASA TM 107121, December, 1995, 26 pages.

A hybrid grid system that combines the Chimera overset grid scheme and an unstructured grid method is developed to study fluid and heat transfer problems. With the proposed method, the solid structural region, in which only the heat conduction is considered, can be easily represented using an unstructured grid method. As for the fluid flow region external to the solid material, the Chimera overset grid scheme has been shown to be very flexible and efficient in resolving complex configurations. The numerical analyses require the flow field solution and material thermal response to be obtained simultaneously. A continuous transfer of temperature and heat

flux is specified at the interface, which connects the solid structure and the fluid flow as an integral system. Numerical results are compared with analytical and experimental data for a flat plate and a C3X cooled turbine cascade. A simplified drum-disk system is also simulated to show the effectiveness of this hybrid grid system.

Steinthorsson, E.; Modiano, David (ICOMP): "Advanced Methodology for Simulation of Complex Flows Using Structured Grid Systems", ICOMP Report 95-28, May, 1995, 14 pages.

Detailed simulations of viscous flows in complicated geometries pose a significant challenge to current capabilities of Computational Fluid Dynamics (CFD). To enable routine application of CFD to this class of problems, advanced methodologies are required that employ (a) automated grid generation, (b) adaptivity, (c) accurate discretizations and efficient solvers, and (d) advanced software techniques. Each of these ingredients contributes to increased accuracy, efficiency (in terms of human effort and computer time), and/or reliability of CFD software. In the long run, methodologies employing structured grid systems will remain a viable choice for routine simulation of flows in complex geometries only if genuinely automatic grid generation techniques for structured grids can be developed and if adaptivity is employed more routinely. More research in both these areas is urgently needed.

Seminars

1995 SEMINARS

Chang, Sin-Chung (NASA Lewis): "New Developments in the Method of Space-Time Conservation Element and Solution Element-Applications to Two-Dimensional Time-Marching Problems"

A new numerical discretization method for solving conservation laws is being developed. This new approach differs substantially in both concept and methodology from the well-established methods- i.e., finite difference, finite volume, finite element and spectral methods. It is motivated by several important physical/numerical considerations and designed to avoid several key limitations of the above traditional methods.

As a part of its design, the new method also strives for *simplicity, generality, and accuracy*. Although it does not use (i) any characteristics-based techniques, (ii) any mesh refinement techniques, (iii) any moving meshes, or (iv) any ad hoc techniques that are used only in the neighborhood of a discontinuity, an Euler time-marching solver developed using the new method is capable of generating highly accurate solutions for a 2-D shock reflection problem used by Helen Yee and others. Specifically, *both the incident and the reflected shocks can be resolved by a single data point without the presence of numerical oscillations near the discontinuity*.

In sharp contrast to the new method, modern high-resolution upwind methods are heavily dependent on characteristics-based techniques. For the 1-D time-dependent case, the characteristics are curves in space-time, and the coefficient matrix associated with the Euler equations also can be diagonalized easily. As a result, these techniques are easy to apply. However, for multidimensional cases, the characteristics are 2-D or 3-D surfaces in space-time. Moreover, the coefficient matrices cannot be diagonalized simultaneously by the same matrix. Because of these complexities, application of upwind methods to multidimensional problems is much more difficult than the new method. Furthermore, upwind methods generally require the use of ad hoc techniques near a discontinuity. Not only is it difficult to apply these techniques in a space of higher dimension, but their application may lead to numerical dissipation which varies from one place to another, and from one Fourier component to another. In other words, numerical solutions may suffer annihilation of sharply different degrees at different locations and different frequencies.

Gazaix, Michel (ONERA, France): "Comparative Study of Data-Parallel and Message Passing Technologies on Distributed Memory Parallel Computers"

In this talk, we present an overview of some work in progress at ONERA in parallel CFD. The two essential modes of parallel programming, Data Parallel and Message-passing, are discussed, with emphasis in practical applications. The performances of several parallel architectures are compared: the Connection Machine CM5, the INTEL PARAGON, the IBM SP1 and the CRAY T3D. We discuss the advantages and drawbacks of these architectures, with respect to traditional well-proven shared-memory vector computers, such as CRAY YMP. Realistic viscous compressible flow computations have been done with FLU3M, a multi-block, finite-volume, structured code, based on an upwind formulation with MUSCL approach. An implicit integration in time is performed in each block, with explicit exchange of two rows of data at the boundaries of the blocks. The problem of the degradation of the convergence to steady state brought by this explicit coupling will be addressed.

Goldman, Louis (NASA Lewis): "Experimental Measurement of Reynolds Stress in a Turbine Stator Wake using Laser Anemometry: Part II"

Last year I presented an experimental method for determining the complete velocity vector and Reynolds stress tensor using laser anemometry (LA). The nonintrusive measurement of the velocity field yields, as an added bonus, turbulence information even a single component LA system. The method I outlined involved the use of least-squares techniques to obtain the three velocity components and the six Reynolds stress parameters, as well as, a statistical estimation of their uncertainties. Preliminary experimental results that I showed at that time were discouraging. I indicated that I hoped to be able to come back and present better measurements. I believe that this is now possible.

In the present talk, I will briefly summarize my previous work for those persons who were not present at my first talk. I will then show the new results I obtained in the stator wake 5% downstream of the vane trailing-edge at the mean radius. Qualitatively, the trends in the experimental data are in agreement with the expectations. For example, the radial velocity component measured was small directed towards the vane hub. This is in accordance with the constant section vane profile, cylindrical endwalls, and the cascade radial pressure gradient. The measured turbulence was anisotropic in the wake or viscous region, but approached isotropic conditions in the "inviscid" region. In addition, the spatial trends for the shear stresses in the wake are similar to previous hot wire measurements for other vane geometries.

Finally, I will briefly discuss my future experimental plans.

Jiang, Bo-nan (ICOMP): "Survey of Recent Advances in the Least-Squares Finite Element Method"

The Galerkin finite element method for selfadjoint elliptic differential equations proved to be optimal and became the dominant computational technique in solid mechanics and other similar field problems. However, attempts to apply the Galerkin approach to non-selfadjoint equations in fluid dynamics and other transport problems encountered serious difficulties. These include: the oscillations caused by first-order convective terms; the instabilities caused by the incompressibility; the discontinuities caused by shocks.

The least-squares finite element method (LSFEM) discussed here is based on minimizing the residuals of the first-order system of differential equations. The LSFEM is a universal method for the numerical solution of all types of partial differential equations without introducing special treatments, such as upwinding, non-equal-order interpolation, staggered grid, operator-splitting, preconditioning, sophisticated shock-capturing, etc. The LSFEM is the simplest method: there is nothing beyond the finite element interpolation, the least-squares principle, the linearization (for non-linear problems) and the time-differencing (for transient problems). The LSFEM leads to discrete problems with symmetric and positive definite matrices. The solution of these problems can be accomplished by efficient element-by-element conjugate gradient methods on parallel computers.

In this talk we will briefly review some new advances in theoretical aspects of the LSFEM. We will show new numerical results for a variety of problems, including incompressible inviscid rotational flows, incompressible viscous flows with heat transfer, low-Mach number compressible viscous flows, and electromagnetic problems.

Benjamin, Michael and Harvey, Rex (Parker Hannifin Corporation): "The Pressure Swirl (Simplex) Atomizer in Gas Turbine Engines"

Simplex atomizers are commonly used in gas turbine fuel injectors. They are generally designed using empirical correlations based on atomizers manufactured using conventional metal removal techniques, and hand finishing to provide required spray angles and good spray patterns. New manufacturing methods are now available where control over the smallest details of the design are

SEMINARS

possible. In order to effectively use these new manufacturing techniques, it is necessary to understand the extent of influence that certain geometric parameters and conditions exert on the very thin fluid sheet emanating from an advanced simplex nozzle.

Most simplex design improvements have been made using empirical methods and almost no use of CFD, as empirical methods generally satisfied all the requirements of the engine manufacturers. However, with advanced engines programs (e.g. AST, HSCT) pushing up the turbine inlet temperature, combustor designers are being forced to specify very tight and demanding performance criteria from the atomizers. This allows the designer control of No_x , unburned hydrocarbons, smoke production and pattern factor in the combustor. Hence the development of a better understanding of the detailed fluid mechanics of atomizers is central to the success of such programs.

Note: The presentation will focus on the simplex nozzle with the intention of identifying research areas of mutual interest for ICOMP and Parker Hannifin.

Liou, William (ICOMP): "Transonic Turbulent Flow Predictions with New Two-Equation Turbulence Models"

The development of advanced propulsion systems for future aerospace vehicles requires accurate computational models of turbulence that can be used in the CFD calculation of the individual component flow. The performance of two turbulence models recently developed at CMOTT in the prediction of two transonic benchmark flows will be presented in this talk. The models include a high-Reynolds number and a low-Reynolds number, $\kappa - \epsilon$, two-equation models. Both models satisfy the realizability constraints of the Reynolds stress. The first of the two flows is an internal transonic flow generated by floor-mounted bumps. The flow was observed to be incipiently separated. The flow has been identified as ONERA Bump A in the EUROVAL effort, a program on the validation of CFD codes with special attention paid to the validation of turbulence models. The second configuration is the Bachalo and Johnson case ($M_\infty = 0.875$). The particular flow was selected as a test case for transonic separated flows in the 1980 AFSOR-HTTM-Stanford conference. The two new models have shown improved capability in predicting the transonic flow with shock/turbulent boundary-layer interactions over the existing models tested.

Eiseman, Peter R. and Cheng, Zheming (Program Development Corporation, White Plains, NY): "Automatic Structured Grid Generation"

An overview will be given for multiblock grid generation with automatic zoning. The presenters shall explore the many advantages and benefits of this exciting technology and will also show how to apply it to a number of interesting cases. The GridPro/az3000 code takes surface geometry definitions and patterns of points as its primary input and produces high quality grids as its output.

Georgiadis, Nick (NASA Lewis): "Recent Turbulence Model Upgrades to the NPARC code"

As part of the NPARC (National PARC code) Alliance's efforts to provide a reliable Navier-Stokes solver for the U.S. aeropropulsion community, several improvements have been made in the past two years to the turbulence model capabilities of the NPARC code and more are planned for the future. New models which have been added to NPARC and are currently being used for inlet and nozzle applications are the Baldwin-Barth one-equation and Chien $k-\epsilon$ two equation models. The $k-w$ model of Wilcox was recently installed and is currently being validated. CMOTT has also prepared a subprogram to be used with NPARC that contains advanced two-equation turbulence models and will allow for more turbulence model enhancements in the future. An overview of the turbulence model improvements to

NPARC will be given in this presentation. In addition, application of the new models to inlet and nozzle flows of recent interest will be demonstrated.

Krishnamoorthy, S., Ramaswamy, B. (Rice University) and Joo, S. W. (Wayne State University): "Instabilities and Nonlinear Flow Development in Thin-Film Flows with Heat Transfer: a Full-scale Direct Numerical Simulation"

The flow of thin liquid film on an incline has many important applications in material processing, biomedical engineering and nuclear industry. Liquid layers can become unstable due to various hydrodynamic instability mechanisms. Theoretical studies based on linear/nonlinear analysis give useful information on critical parameters such as film thickness and wavenumber of the disturbance. However, they can not follow the dynamics of the flow over a long period of time when nonlinearity becomes significant. To overcome this difficulty and to understand comprehensively the competition among various forces acting in the system, governing equations of conservation of mass, momentum and energy along with kinematic equation for free-surface motion are solved using finite element method. An arbitrary Lagrangian Eulerian frame of reference is used to update precisely the motion of the interface. Rupture dynamics and rivulet formation has been analyzed for various Reynolds numbers based on the film thickness.

Pletcher, Richard H. (Iowa State University): "Recent Results on Large-Eddy Simulation and Time-Accurate, Preconditioned Algorithms and Multigrid Acceleration"

Progress on some studies underway at Iowa State University will be discussed. The overall objective of the research is to move large-eddy simulation technology toward more applied configurations and problems. An all-speed strategy is being followed in the numerical simulations which permits the same general methodology to be applied to fully incompressible flows, compressible flows at very low Mach numbers where effects of property variations need to be accounted for, and at transonic and supersonic speeds. This flexibility is achieved by employing a coupled, compressible formulation with low Mach number preconditioning. Aspects of algorithms will be discussed including the effectiveness of multigrid acceleration as a strategy for reducing the CPU time required for unsteady simulations. Large-eddy simulation results for several flows obtained with a dynamic subgrid-scale model will be presented including a channel heated and cooled with a wall temperature ratio of 3.0.

Colella, Phillip (The University of California, Berkeley and Lawrence Livermore National Laboratory): "High-resolution Numerical Methods for Low-Mach Number Flows"

In this talk, we will discuss some approaches to designing numerical methods for low-Mach number, advectively-dominated time-dependent fluid flows. The approach is based on two ideas: (1) the use of low-Mach number asymptotics to eliminate acoustic waves, and of the Hodge/Helmholtz projection to express the resulting elliptic constraint equations; and (2) the use of predictor-corrector time discretization to split the equations into hyperbolic and second-order elliptic/parabolic terms, with appropriate strategies for discretizing each. We will present examples from incompressible and nearly incompressible flow, combustion, and flows in irregular geometries.

Yao, Minwu (Ohio Aerospace Institute): "Large Extensional Deformations of Viscoelastic Liquid Bridges in Filament Stretching Devices"

The prevalence of extensional flows in industrial polymeric material processing has motivated the investigation of the rheological properties of non-Newtonian fluids under idealized extensional flow

SEMINARS

conditions. Filament stretching device is one of the rheometers which have been proposed as potential methods for measuring transient extensional viscosities of viscoelastic polymer solutions. Although the kinematics in this device approximate ideal uniaxial elongation, the presence of a deformable free surface and two rigid, non-deforming end-plates result in an appreciable shear component in the deformation history. This modifies the elongational stresses that develop in the fluid and complicates the direct calculation of the extensional (or Trouton) viscosity from the measured axial force history. Due to the large extensional deformation and the viscoelastic behavior of the liquid, the direct, full-scale analysis of the liquid bridge in filament stretching device poses a great challenge to the CFD world and there is a general no closed-form solution nor adequate published results available in the literature.

This research represents a preliminary study forming part of the support modeling effort for a designated NASA space experiment. To model the extraordinary extensional deformation, a three-stage approach is proposed. The first stage studies the initial motion of the liquid bridge using an approximate analytical solution obtainable under certain assumptions. The second stage covers the deformation range which is numerically accessible based on the current finite element technique and a new remeshing technique developed in Nekton. The last stage studies the asymptotic behavior at larger deformation by utilizing the Cosserat 1-D model. The simulated transient extensional viscosity (or Trouton ratio) is compared with theory and the available experimental data. The idea of reduced diameter device and its effects on the measurement of extensional viscosity are also studied through full-scale simulation.

Moon, Young J. (Korea University): "Development of Unstructured Navier-Stokes Solvers for Compressible and Incompressible Flows"

In this talk experiences in developing unstructured Navier-Stokes solvers on a triangular adaptive mesh will be presented for both compressible and incompressible flows.

Solution accuracy of unstructured Navier-Stokes solvers on a triangular mesh will be discussed, by comparing results with some benchmark solutions of compressible and incompressible flows. Convergence behavior of the unstructured solver are also examined using different multi-stage Runge-Kutta schemes. Finally, applications of the unstructured adaptive mesh algorithm will be presented for various compressible and incompressible flows.

Hunt, Barry (Cincinnati, Ohio): "GENESIS: A Boundary Integral Approach to Multi-Disciplinary, Nonlinear Physics (with Particular Emphasis on Aeroacoustics)"

Panel methods were for many years the mainstay tool for aerodynamic design, though the underlying mathematics was limited to the linear regime. These techniques went out of fashion as Computational Fluid Dynamics (CFD) tools based on *differential* calculus emerged, capable of solving nonlinear problems. As industry moves toward large interdisciplinary problems, the inherent deficiencies of differential methods are becoming increasingly apparent -- especially for **aeroacoustics**. If multidisciplinary simulation and optimization are to become a reality, an alternative approach to CFD is required.

GENESIS is an emerging *boundary integral* technology aimed at general, nonlinear problems of computational physics, representing real, multidisciplinary, engineering systems, with computation and data-preparation times *orders less* than current CFD techniques, without the need for either complicated meshes or supercomputers. It forms a natural extension to the panel codes which for decades have been in daily use by the entire aerospace community for (linear) predictions of full aircraft configurations.

The seminar shows how the *integral* calculus of panel methods extends to fully nonlinear problems (e.g. Unsteady, transonic flows). *GENESIS* exploits a mathematical vector identity derived from the theorems of Green and Gauss, allowing the effects of *field* nonlinearities (compressibility,

rotationality) to be translated to the body *boundary*, and treated as iterative modifications to the boundary conditions of an equivalent pseudo-linear problem; the complex meshes of traditional CFD are thus unnecessary. The approach unifies the mathematics of elliptic, hyperbolic and parabolic systems, automatically taking account of signal directionality in supersonic regions of a flow, and rendering techniques such as "artificial viscosity" and "upwinding" unnecessary. Two complementary methods will be described: "**Semi-Analytic Methods**" which *analytically* (piecewise) solve an augmented form of the Euler equation, while "**Residual Methods**" solve the residual field defining the difference between the augmented and true Euler equations. Their combination yields the Euler solution, with exact shock discontinuities and downstream entropy flux (vorticity).

The iterative definition of the boundary conditions fits in naturally with the linear solver "**SAVER**" upon which *GENESIS* is built. *SAVER* replaces the matrix techniques employed in panel methods with a relaxation cycle which is fast, robust, accurate, and equally suitable for *analysis* and *design*. Nonlinear, steady-state problems take only fractionally longer to solve than linear problems, which are themselves solved faster by *SAVER* than by any known method. This relaxation approach immediately opens up the possibility of affordable solutions of *evolutionary* problems -- in particular, *aeroacoustics* -- for complete aircraft configurations.

The seminar reviews the mathematical basis of *GENESIS* and its linear subset, *SAVER*, contrasting this with earlier panel method formulations. It also presents some recent nonlinear (steady-state, transonic) results, and discusses the diversification of *GENESIS* into other branches of computational physics. Particular emphasis will be placed on the perhaps unfamiliar (integral) mathematical and physical concepts which allow unsteady problems to be treated in the same framework as steady-state problems.

Ta'asan, Shlomo (Carnegie Mellon University): "Essentially Optimal Multigrid Method for Steady-State Euler Equations"

In this talk we discuss a novel approach for the solution of inviscid flow problems for subsonic compressible flows. The approach is based on canonical forms of the equations, in which subsystems governed by hyperbolic operators are separated from those governed by elliptic ones. The discretizations used as well as the iterative techniques for the different subsystems, are inherently different.

Hyperbolic parts, which describe, in general, propagation phenomena, are discretized using upwind schemes and are solved by marching techniques. Elliptic parts, which are directionally unbiased, are discretized using h-elliptic central discretizations, and are solved by pointwise relaxations together with coarse grid acceleration. The resulting discretization schemes introduce artificial viscosity only for the hyperbolic parts of the system; thus a smaller total artificial viscosity is used, while the multigrid solvers used are much more efficient. Solutions of the subsonic compressible Euler equations are achieved at the same efficiency as the full potential equation.

ALLSPD-3D Combustor Code Workshop

1995 ALLSPD-3D Combustor Code Workshop

November 14-15, 1995

Objective and Scope

The purpose of this Workshop is to release Version 1.0 of the ALLSPD-3D Combustor Code, a numerical tool developed at the NASA Lewis Research Center for simulating chemically reacting flows in aerospace propulsion systems. It provides the designer of advanced engines an analysis tool that employs state-of-the-art computational technology. The code is capable of calculating multiphase, swirling flows over a wide Mach-number range in combustors of complex geometry.

ALLSPD-3D has evolved from the two-dimensional code ALLSPD, which was released in June, 1993. Besides extension to three dimensions, the new code features several improvements and enhancements, including a user-friendly GUI, multi-platform capability (supercomputers, workstations, and parallel processors), improved turbulence and spray models, and more generalized chemistry.

The Workshop will present information relating to ALLSPD-3D's numerics, physical models, code structure, and test cases. Workshop participants will receive a copy of ALLSPD-3D, Version 1.0, a user manual, and other documentation.

Although the complete Workshop will last a day and a half, it has been designed in such a way that a person who can only attend the first day will receive the essential information. The morning session of the second day will provide interested parties the opportunity for interaction with the ALLSPD-3D team and hands-on experience in running the code.

This workshop is cosponsored by NASA Lewis Research Center and ICOMP, and will be held at the Ohio Aerospace Institute, located adjacent to the Center.

Organizing Committee

WORKSHOP CHAIRMAN

Robert Stubbs (NASA LeRC)

INDUSTRY

Beverly Duncan (NYMA, Inc.)

Angela Quealy (NYMA, Inc.)

Jinho Lee (NYMA, Inc.)

UNIVERSITY

Kuo-Huey Chen (University of Toledo)

NASA

Jeffrey Moder (LeRC)

David Fricker (LeRC)

Edward Mularz (LeRC)

ICOMP

T. Keith (OAI/University of Toledo)

Workshop Agenda

Welcome by

Louis Povinelli, Acting Chief of Internal Fluid Dynamics Division

Numerical Algorithms and Physical Models

Test Cases - ALLSPD Team

- Laminar and turbulent non-reacting flows
- Laminar spray combustion flows
- Turbulent premixed combustion flows
- Turbulent spray combustion flows

Parallel Implementation

Code Structure, I/O, GUI

Future Upgrades

Audience Feedback

Distribution of ALLSPD-3D Code

- Open Discussion
- Demonstrations

Informal “hands-on” Session

AST Engine Noise Workshop

AST Engine Noise Workshop December 12-14, 1995

Objective

The purpose of this workshop is to review the technical work that is being done for engine noise reduction in NASA's Advanced Subsonic Technology (AST) Noise Reduction Program. Presentations will be given by NASA researchers and industry/university principle investigators who have contracts or grants supported by the program. Work in jet noise, fan noise, linear technology, and active noise control will be presented, which is part of the Engine Noise Reduction and Nacelle Aeroacoustics subelements of the AST Program.

One of the primary objectives of the AST Program is to give U.S. Aeronautics industry a competitive advantage in the international marketplace. Therefore, non-disclosure agreements will be required for university and industry participants who are not members of the AST Noise Reduction Technical Working Group.

There will be parallel sessions during each day of the workshop. This workshop is cosponsored by NASA Lewis Research Center and NASA Langley Research Center, and will be held at the Ohio Aerospace Institute, which is located adjacent to NASA Lewis Research Center.

Workshop Agenda

Welcome by

Pete Batterton, AST Program Manager, NASA LeRC

William Willshire, AST Noise Reduction Program Manager, NASA LaRC

Fan Experiments and Broadband Noise (Auditorium)

Session Chairman: Dennis Huff, NASA LeRC

"Current Fan Noise Testing in the NASA Lewis 9x15 Wind Tunnel" James Dittmar, NASA Lewis, and Larry Bock, Pratt & Whitney

"Comprehensive Measurement of the Far Field of the Langley 12-Inch ADP Fan" Russ Thomas, VCES

"Application of Inlet Bleed Concept to the Langley 12-Inch Advanced Ducted Propeller Simulator" Lorenzo R. Clark, NASA Langley

"Wake Measurement Strategies for Reduction of Turbomachinery Fan Noise" Ian A. Waitz, MIT

"Comparison of Scale Model Fan Noise Characteristics in Different Wind Tunnel Facilities" P. Y. Ho, \AA

"Boeing 18-Inch Fan Test - Effects of Boundary Layer, Tip Clearance, Loading, and Stator on Broadband Noise" Paul Joppa, Boeing

"Boeing 18-Inch Fan Test - Identification and Removal of Background-Noise Contamination" Dan Scharpf, Boeing

"Boeing 18-Inch Fan Test - Effects of Boundary Layer, Tip Clearance, Loading and Stators on Performance" Tim Patten, Boeing

"Boeing 18-Inch Fan Test - Effects of Boundary Layer, Tip Clearance and Loading on Unsteady Flow" Ulrich Ganz, Boeing

"Broadband Noise from Subsonic Rotors" Ramani Mani, GE CR&D

"Cascade Effects of the Generation of Broadband Fan Noise" Stewart A. L. Gregg, Florida Atlantic University

Jet Noise (Library)

Session Chairman: Eugene Krejsa, NASA LeRC

"Internal Mixers for Low Bypass Ratio Engines" Naseem Saiyed, NASA Lewis, Paul Montuori, Pratt & Whitney

"Subsonic Jet Noise Technology Development at GEAE" Muni Majjigi and Bob Babbitt, GE

"Forced Mixer Nozzle Optimization Status" Don Weir, Allied Signal

"Status of Industry/NASA Langley JNL Jet Noise Data Base and Suppression Program for Separate Flow Nozzles" Jack Seiner, NASA Langley

"A Three-Microphone Technique to Separate Jet Noise from Upstream Noise and Other Attempts of Locating Jet Noise Sources" K. K. Ahuja and K. Massey, Georgia Institute of Technology

"Mixing Noise from Non-Circular Jets" Abbas Khavaran, NYMA, Inc.

"An Assessment of the MGB Jet Noise Analysis for Multi-Stream Axisymmetric and Mixer Lobed Nozzles" Thomas J. Barber, UTRC

"Computation of the Effects of Flow on Noise from Subsonic Jets" Christopher Tam, Florida State University

"Acoustic Duct Modes Excited by an Axially Convecting Noise Source in Sheared Flow" Vinod G. Mengle, Allison

Fan Noise Analysis (Auditorium)

Session Chairman: John Groeneweg, NASA LeRC

"Progress in Development of Coupled Fan Noise Prediction System" Don Hanson and Dave Topol, Pratt & Whitney, Harold Meyer, Hamilton Standard, and Ken Kousen, UTRC

"Influence of Vane Sweep & Lean on Rotor-Stator Interaction Tone Noise" Ed Envia, NYMA, Inc.

"Quiet High Speed Fan Program Status" Don Weir, Allied Signal

"Computations of Acoustic Model Propagation in a Duct with a Rotating Grid Block" Robert T. Biedron, AS&M, Inc.

"Research Aspects Involved with UNCLE-TURBO Flow Code Development" Mark Janus, Mississippi State University

"Unsteady Aerodynamic Analyses for Blade Rows" Joe Verdon, UTRC

"Aeroacoustics of High Speed Fans in Nonuniform and Swirling Flows" H. M. Atassi, University of Notre Dame

"Sensitivity Analysis and Design Strategies for Three-Dimensional Aeroacoustic Flows in Turbomachinery" Kenneth C. Hall, Duke University

"Prediction of Wakes in Turbomachinery Flows Using Three-Turbulence Models" Aamir Shabbir and Jiang Zhu, ICOMP and Mark Celestina, NYMA

"ANOPP Updates for Noise Predictions" Robert A. Golub, NASA Langley

"Active Control of Inlet Noise Through Boundary-Layer Modulation" Anders O. Anderson, Boeing

Liner Technology (Library)

Session Chairman: Joe Posey, NASA LaRC

"Nacelle Acoustic Liners Designed for Broadband Attenuation" Jerry Bielak, Boeing

"Acoustic Behavior of a Novel Viscously Controlled Liner" Alan S. Hersh, Hersh Acoustical Engineering, Inc.

"Acoustic Properties of Liners with Linearly Varying Cavity Depths" Michael Jones, Lockheed

"Acoustic Properties of Liners with Harmonically-Tuned Parallel Elements" Sharon E. Tanner, NASA Langley

"A Theoretical Approach to Nonlinear Liner Optimization" William E. Zorumski, NASA Langley

"Full-Scale and Sub-Scale Treatment Suppression Comparisons for an Engine Inlet" R. E. Kraft, GE

"UPS vs. Engine Treatment Performance Comparisons" P. Y. Ho and P. R. Gliebe, GE

"Comparison of Predicted and Measured Impedance for Sub-Scale Treatment Panels" Jia Yu, Rohr, Inc. and R. Kraft, GE

"A Numerical Method for Extracting Liner Impedance From Measured Data" Willie R. Watson, NASA Langley

"A Unique Test Facility to Measure Liner Properties in the Presence of Cold and Heated Flows and a New Broadband Liner" K. K. Ahuja and R. J. Gaeta, Georgia Institute of Technology

"Some Comments on Impedance Measurement Technology" Tony L. Parrott, NASA Langley

"A Smart Active Acoustic Liner and its Application for Aircraft Engine Bypass Fan Ducts" Istvan L. Ver, BBN Acoustic Technologies

Fan Noise Analysis and Liner Technology (Auditorium)

Session Chairman: Tony Parrott, NASA LaRC

"AFT Fan Duct Acoustic Radiation" W. Eversman, University of Missouri-Rolla

"Computational Method Development for Turbofan Acoustics" John E. Caruthers, The University of Tenn Space Institute

"Computation of Noise Radiation from Turbofans" M. Nallasamy, NYMA, Inc.

"Boundary Integral Methods for Calculation of Ducted Fan Noise Radiation" M. K. Myers, George Washington University/JIAFS

"Application of BIEM to Ducted Fan Noise Prediction" Feri Farassat, NASA Langley

"Computational Aeroacoustics of Engine Inlets" Lyle N. Long and Yusuf Ozyoruk, Penn State University

"Ray Tracing for Three-Dimensional Nacelle Acoustics" Robert P. Dougherty, Boeing

"Advanced Nacelle Liner Development (Bulk, Scale Treatment, New Concepts, Etc.)" Jia Yu, Rohr, Inc.

"22-Inch ADP Fan Rig Lining Design" John Premo and Jerry Bielak, Boeing, John Low and Doug Mathews, Pratt & Whitney

Active Noise Control (Multipurpose Room)

Session Chairman: Larry Heidelberg, NASA LaRC

"The Active Noise Control Fan Rig and an Overview of the LeRC ANC Program" Larry Heidelberg, NASA Lewis

"Potential System Noise Impact of ANC Suppression" R. E. Kraft and B. A. Janardan, GE

"Development of ANC Actuator and Modal Control Systems for Turbofan Application" Frederic Pla and Ziqiang Hu, GE CR&D

"Results of the GE Modal Control System in the NASA ANCF" Daniel L. Sutliff, NASA Lewis (NRC)

"Active Control of Fan Generated Tone Noise" Carl H. Gerhold, NASA Langley

"A Review of Rotor/Stator Active Noise Control at the Source" John Simonich, UTRC

"Active Control of Discrete Frequency Noise Generated by Rotor/Stator Interaction" Scott Sawyer and Sanford Fleeter, Purdue University and John Simonich, UTRC

"Electrostrictive Polyurethane Actuators for the Active Control of Fan Noise" Alan Curtis, BBN Acoustic Technologies

"Active Control Source Cancellation of Multi-Mode Rotor/Stator Interaction Noise" Bruce Walker, Hersh Acoustical Engineering, Inc.

"Active Control Helmholtz Resonator Absorption of Multi-Mode Rotor/Stator Interaction Noise" Bruce Walker, Hersh Acoustical Engineering, Inc.

"Active Control of Inlet Noise From a Turbofan Engine" J. Smith, R. Burdisso and C. Fuller, Virginia Tech

"Analytical Modeling of Active Control of Inlet Noise" R. Burdisso and C. Fuller, Virginia Tech and S. Gregg, Florida Atlantic University

"Concepts for the Control of Broadband Noise from Turbofan Engines" C. R. Fuller and R. Burdisso, Virginia Tech

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE June 1996		3. REPORT TYPE AND DATES COVERED Technical Memorandum
4. TITLE AND SUBTITLE Institute for Computational Mechanics in Propulsion (ICOMP) Tenth Annual Report-1995			5. FUNDING NUMBERS WU-505-90-5K NCC3-370	
6. AUTHOR(S) Theo G. Keith, Jr., Karen Balog, and Louis A. Povinelli, editors				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191			8. PERFORMING ORGANIZATION REPORT NUMBER E-10302	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, D.C. 20546-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA TM-107248 ICOMP-96-01	
11. SUPPLEMENTARY NOTES ICOMP Program Director, Louis A. Povinelli, organization code 2600, (216) 433-5818.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category 64 This publication is available from the NASA Center for Aerospace Information, (301) 621-0390.			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) The Institute for Computational Mechanics in Propulsion (ICOMP) is operated by the Ohio Aerospace Institute (OAI) and funded under a cooperative agreement by the NASA Lewis Research Center in Cleveland, Ohio. The purpose of ICOMP is to develop techniques to improve problem-solving capabilities in all aspects of computational mechanics related to propulsion. This report describes the activities at ICOMP during 1995.				
14. SUBJECT TERMS Numerical analysis; Computer science; Mathematics; Fluid mechanics			15. NUMBER OF PAGES 63	
			16. PRICE CODE A04	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	

**National Aeronautics and
Space Administration**

Lewis Research Center
Cleveland, OH 44135-3191
ICOMP 0A1

Official Business

Penalty for Private Use \$300